6 Groundwater

ROYAL HASKONING

Chapter 6: Groundwater

6.1 Supported models....2-3
6.2 Walk through example with FLAIRS (tutorial)....2-4
6.2.1 Definition of a Triwaco Project....2-5
6.2.2 Setting up a groundwater model....2-6
6.2.3 Setting up a discretisation dataset (calculation grid) 2-7
6.2.4 Step 1: Creating a discretisation dataset....2-8
6.2.5 Step 2: Defining the model boundary (define a vector map with DIGEDIT)....2-10
6.2.6 Step 3: Defining the position of watercourses, fixed lines (define a vector map, shape file) ...2-11
6.2.7 Step 4: Defining the position of sources, fixed nodes (define a vector map, shape file) 2-13
6.2.8 Step 5: Defining the position of node density areas (define a vector map with DIGEDIT)....2-14
6.2.9 Step 6: Generating the grid....2-15
6.3 Setting up design dataset, the conceptual model set up 2-17
6.3.1 Creating a Design data set....2-17
6.3.2 Input of parameters covering the whole model area (type 'node') 2-19
6.3.3 Input of source data (type 'source')....2-22
6.3.4 Input of parameters for watercourses (type 'river') 2-24
6.3.5 Input of boundary conditions (type 'boundary')....2-26
6.4 Setting up a simulation data set (first simulation)....2-28
6.4.1 Creating a calibration data set....2-28
6.4.2 Allocation of parameters defined by a constant, vector map, raster map or table to the grid...2-30
6.4.3 Definition and allocation of an expression to the grid....2-31
6.4.4 Allocating an entire dataset (Build) making all parameters up-to-date....2-32
6.4.5 Viewing and checking allocated data (in Triwaco and TRIPLOT)....2-32
6.4.6 First simulation....2-34
6.4.7 Viewing and presenting results....2-35
6.4.8 Make results available in the dataset as parameter....2-36
6.4.9 Speed up the calculation time of a simulation (extra)....2-36
6.4.10 Comparing simulation results with measurements (calibration)....2-37
6.5 Setting up a Scenario data set....2-39
6.5.1 Creating a scenario dataset....2-39
6.5.2 Create a scenario by modifying a parameter 2-40
6.5.3 Run the Scenario simulation....2-41
6.5.4 Combining and processing of model output and parameters....2-41
6.6 Setting up a Transient data set....2-44
6.6.1 Creating a Transient dataset....2-44
6.6.2 Input of transient parameters constant in time (initial head, storage coefficient and porosity)..2-46
6.6.3 Input of specified stress periods (abstraction well)....2-47
6.6.4 Input is variable through time, definition by time series (precipitation excess)....2-48
6.6.5 Transient simulation....2-50
6.6.6 Viewing results....2-51

6.1 Supported models

The following groundwater models are supported by the triwaco modelling environment:

Modelcode	Developed by	Description	Chapter
FLAIRS	Royal Haskoning	Finite element steady-state and transient groundwater flow for multi-layered systems	6.3
FLAIRS-VD	Royal Haskoning	Finite element steady-state and transient groundwater flow for multi-layered systems accounting for variable density of groundwater (as a boundary condition)	6.3.?
MODFLOW-96/2000	USGS	Finite difference steady-state and transient groundwater flow for multi-layered systems	6.4
SEAWAT	USGS	Finite difference steady-state and transient variable-density, groundwater flow and solute-transport for multi-layered systems three-dimensional (combinations of MODFLOW and MT3D)	6.5

6.2 Walk through example with FLAIRS (tutorial)

There are several possibilities to get to know **Triwaco**. The most extensive information on the software package can be found in the next chapters of the manual, that includes not only an explanation of how to run the software, but also contains extensive background information of the different modules and supported model codes. This information can also be accessed by the Help function.

This tutorial gives an introduction on how to set up and run a groundwater model in **Triwaco**. It is meant for those who are familiar with groundwater modelling and wish to get a quick view of the normal method to set up and to run a groundwater model, and the standard possibilities of **Triwaco**. A complete view is obtained by using the manual.

It is strongly recommended that prior to starting this exercise one first reads through the previous chapters which explain the general philosophy and handling of the Triwaco modelling environment. Especially chapters 3, 4 and 5 is recommended to read first.

The model set up in this tutorial will be located in the directory C:\My Model\TutorialProject1\ . All data referred to in the text is available in the directory C:\My Models\TutorialData\. A resulting version of the model is located in the directory C:\My Model\Tutorial\. So when things go wrong or you don't know what to do one can always refer to this working model. Below an overview of the successive steps of building a model in Triwaco is given.

6.2.1 Setting up a triwaco project
6.2.2 Setting up a groundwater model
6.2.3 Setting up a discretisation dataset
6.2.4 Setting up a design dataset
6.2.5 Setting up a simulation dataset (steady-state)
6.2.6 Setting up a simulation dataset (transient)
6.2.7 Setting up a scenario dataset

Additional steps may include solute transport, transient or other calculations. In this tutorial one often occurring additional calculations will be explained.

6.2.8 Setting up an effect model
6.2.9 Setting up a pathline dataset

Model building starts with the choice of boundaries and collection of data. This is done without the use of the software, and will not be discussed here. We advise you to make a topographical background map that will help you orientate while using the digitising and presentation modules of **triwaco**. A background map can have different formats (such as a picture (*.bmp, *.jpg), ArcInfo-ungenerated (*.ung) and DXF).

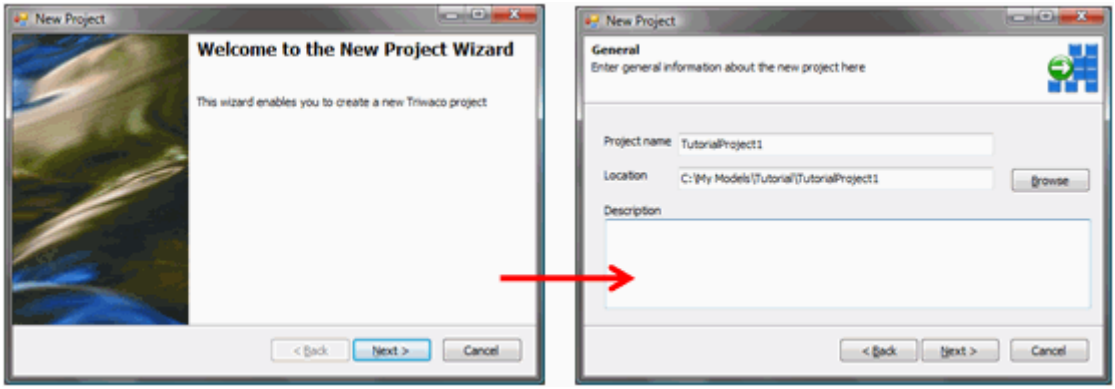
An example of a background map is available after installing **triwaco** on your hard drive.
The file is located in the directory My Models\TutorialData\topo.bmp. The bitmapfile can be used in either the editor DIGEDIT or the viewer TRIPLOT.

triwaco works with a clear hierarchical data storage structure. The entry always is a project that can contain several models. Every model consists of different connected datasets that contain different parameters.

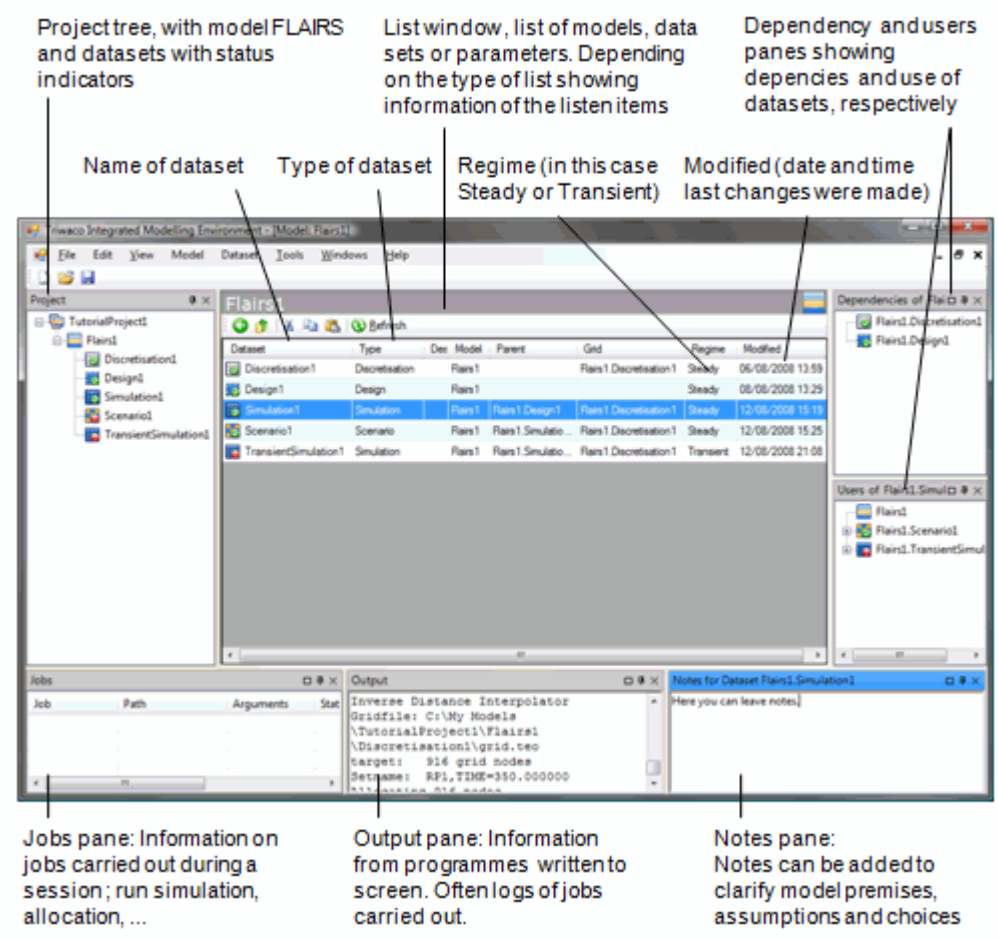
6.2.1 Definition of a Triwaco Project

Now we show you the steps to take for making a groundwater model using **Triwaco**. We keep to the 'main route'; extra options are mentioned with the letter **E** and shown in *italic*. Important notices are indicated with **NB**.

Modelling with **Triwaco** always starts by defining a project. 'File' 'New' if you set up a new modelling project (otherwise 'Open' - and look for the name of your project). A wizard will pop up which will guide you through setting up de project.



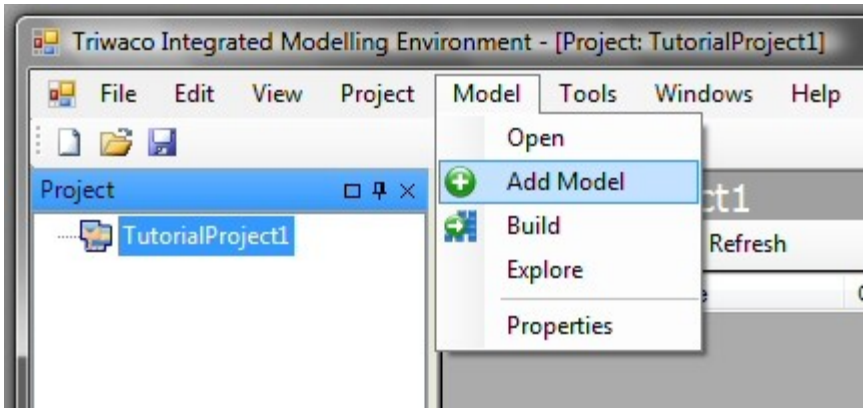
You now see the start window of the modelling environment. It gives all information of the modelling project, models, datasets and parameters. For more information on the modelling environment see chapter 5. In the figure below some important information is given about what can be displayed when working with the modelling environment Triwaco. To add or remove windows go to 'view' in one of the pull down menus. The modelling environment is fully customizable and each window can be put at any location by simply drag and drop.


Project tree, with model FLAIRS and datasets with status indicators
List window, list of models, data sets or parameters. Depending on the type of list showing information of the listen items
Dependency and users panes showing depencies and use of datasets, respectively
Name of dataset
Type of dataset
Regime (in this case Steady or Transient)
Modified (date and time last changes were made)
Jobs pane: Information on jobs carried out during a session; run simulation, allocation, ...
Output pane: Information from programmes written to screen. Often logs of jobs carried out.
Notes pane: Notes can be added to clarify model premises, assumptions and choices

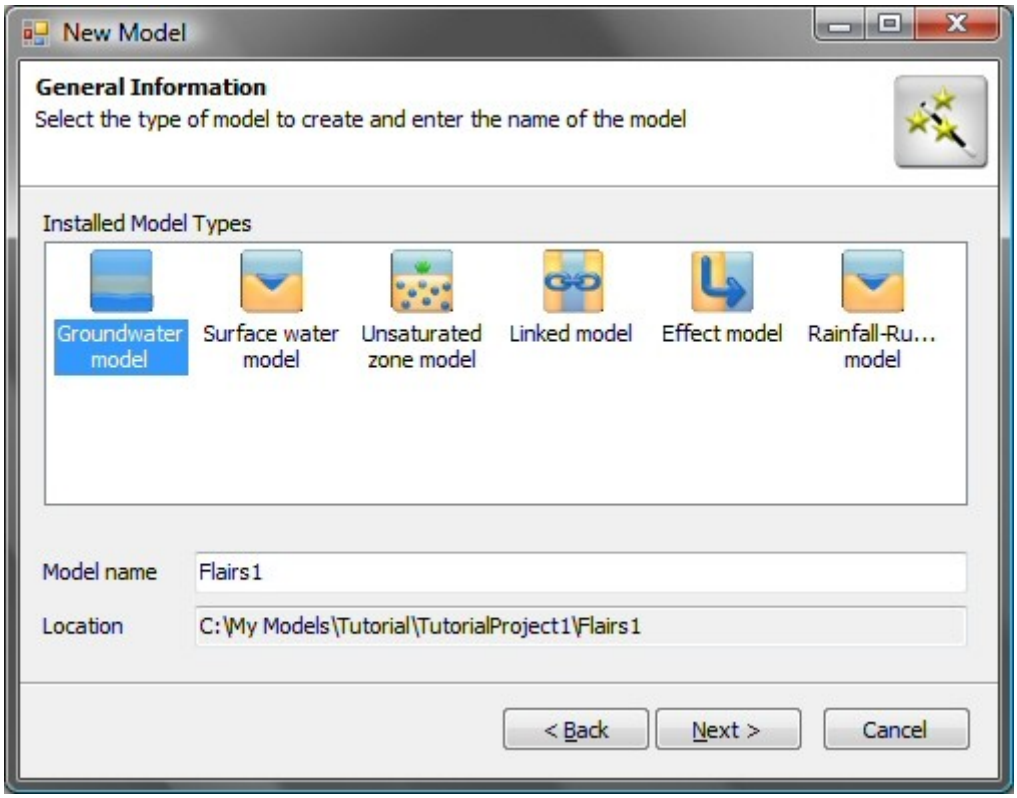
- *It is possible to use your favourite text editor, instead of the standard OpenSource editor that comes with Triwaco, Notepad++. To do this go to 'Tools' in the task bar at the top. Select 'Edit Database'. This will open the database in Access or other database editor. Go to Applications and change the path and location of your favourite text-editor.*

6.2.2 Setting up a groundwater model

Adding a groundwater model to the project is done by: '*double click* TutorialProject' 'Model' 'Add Model', or by '*right mouse click* TutorialProject' 'Add Model' if you set up a new model (otherwise an existing model will be listed in the project).



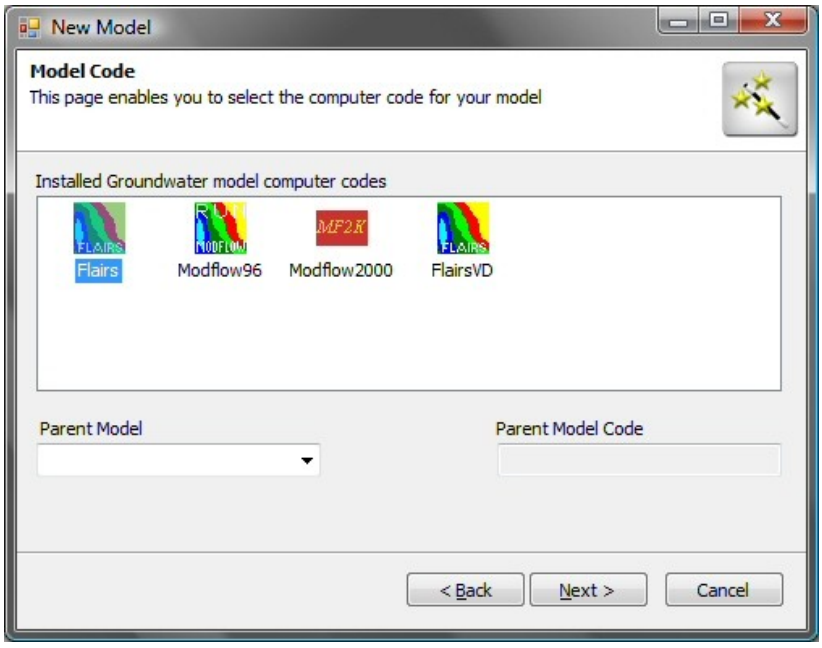
A wizard will pop up. In the first window select 'Next' to continue. In the second window one can choose the type of model. In our case we want to set up a 'Groundwater model'. So select Groundwater model and give it the name Flairs1 (since we will be using the model code Flairs).



In the next window one can select the model code to be used in the simulation. Currently **Triwaco** supports FLAIRS (a finite element code) and a special version of FLAIRS that supports variable density as a boundary condition, the USGS modelling code MODFLOW-96 and MODFLOW-2000. In this tutorial we will set up a groundwater model with the model code FLAIRS.

Flairs

Flairs calculates the groundwater heads and fluxes in a groundwater domain that is divided into aquifers and aquitards. A so called quasi 3D approach. Important features in **Flairs** are the **rivers** (line-source/sinks) and (point)-**sources**, which are active within aquifers, and the large selection of different **top systems** that control the flux from the surface or confining layer to the first aquifer. Hydrogeological parameters are given at the nodes of a Finite Element Grid.



In the window there is also an option for selecting a parent model. This is used when setting up a scenario based on a previously created model. For now this option is not used. Select 'Next' and 'Finish' to create the model.

The groundwater model with the name Flairs is now added to the project. The groundwater model can be opened in several ways: In the project tree window ('*double click* TurorialProject1' or '*expand* TurorialProject1') or in the project window by opening the context menu and 'Open' or by selecting the pull down menu 'Model' 'Open'.
The model is currently empty and contains no datasets. In the next paragraphs the necessary datasets will be added to run the first simulation. We will start by setting up a discretisation dataset where the calculation grid is defined.

6.2.3 Setting up a discretisation dataset (calculation grid)

For a groundwater model a calculation grid is set up in the following six steps :

1. definition of the discretisation dataset
2. definition of the model boundary
3. definition of surface water or other line elements (faults for instance)
4. definition of the position of sources
5. definition of the node density or cell size

When these items are defined the grid can be generated (step 6).

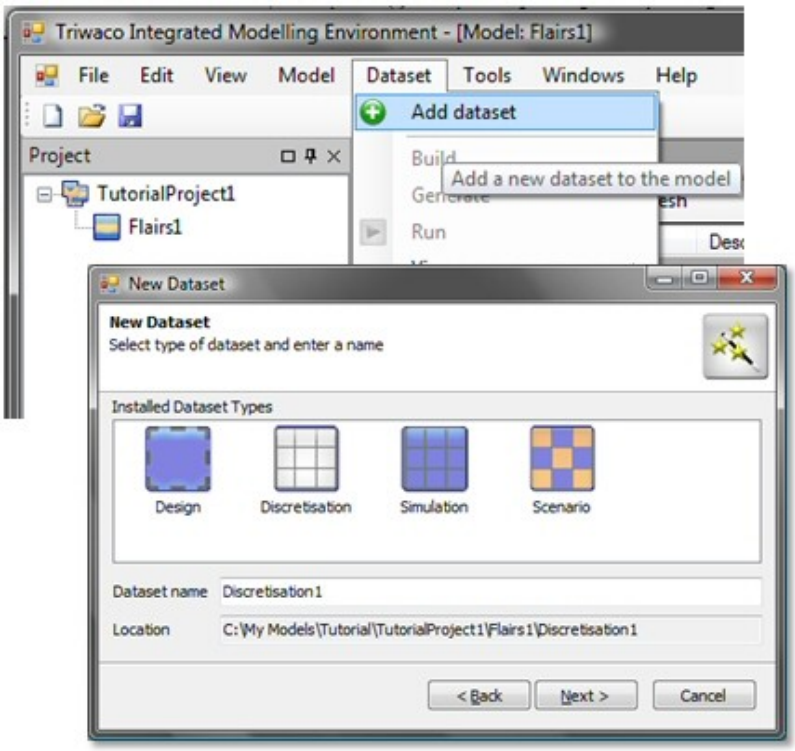
6.2.4 Step 1: Creating a discretisation dataset

The data for the generation of a calculation grid is defined in the discretisation dataset. You'll need: the positions of the boundary, the linear surface water that you want to include in the model and the sources (the grid is taking these elements into account) and specified sub-areas with a certain density of the calculation grid. These data are entered successively. We will introduce the several ways in which Triwaco can handle multiple type file formats. For the discretisation we will use a shape file for the rivers and sources. Both the boundary and density polygons are created by the graphical editor DIGEDIT that comes with Triwaco. All data used in this tutorial is available in the directory My Models\TutorialData\.

Add the dataset: 'Dataset','Add Dataset'. A pup up window will appear similar to that of adding a model. Again in the first window select next to continue. In the second window one can choose the type of dataset. There are four types of datasets each with its own characteristics and purpose:

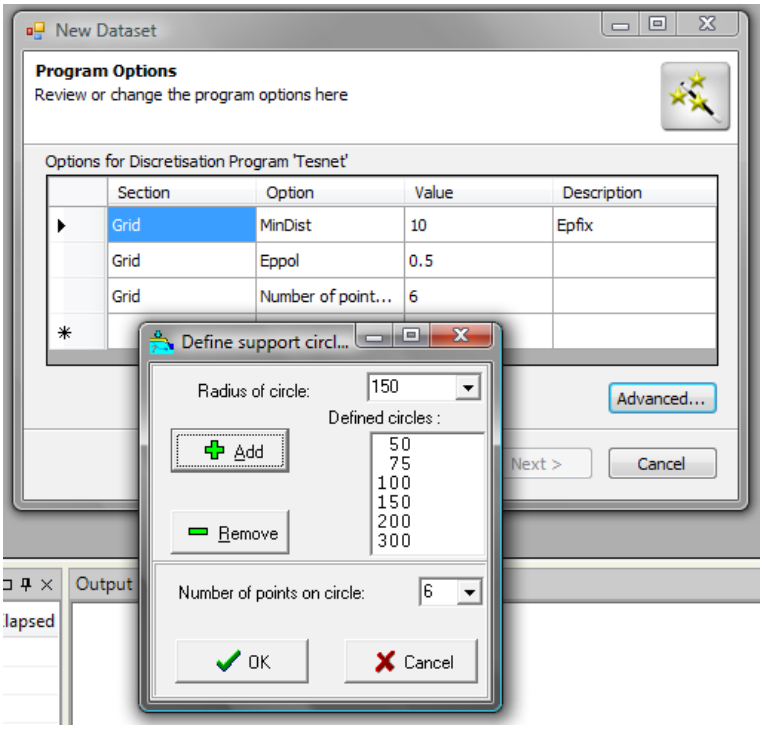
- **Discretisation**: Defines the calculation grid (boundaries and stresses like watercourses and wells)
- **Design**: Defines the conceptual model using GIS maps and tables.
- **Simulation**: Here is where the data from the conceptual model is linked to the calculation grid. The model is now prepared to run with the modelcode.
- **Scenario**: Is similar to the simulation dataset. It is base upon the simulation dataset or another scenario dataset. The dataset is created with parameters linked to the parent dataset. Only parameters that need to be altered for that scenario have to be specified.

Each of these will be created in this tutorial and will make clear the differences between them. For now select 'Discretisation' and select 'Next'.



In the next window on may want to select a parent dataset. This is only of importance when a new grid is created based on an existing dicretisation dataset (like a scenario). For instance for telescoping / window model or simply changing the model boundary or stresses. The nice thing about this option is that a new model can be creating based on an existing conceptual model. For now we leave this option and select 'Next'. The following window appears for the definition of program options.

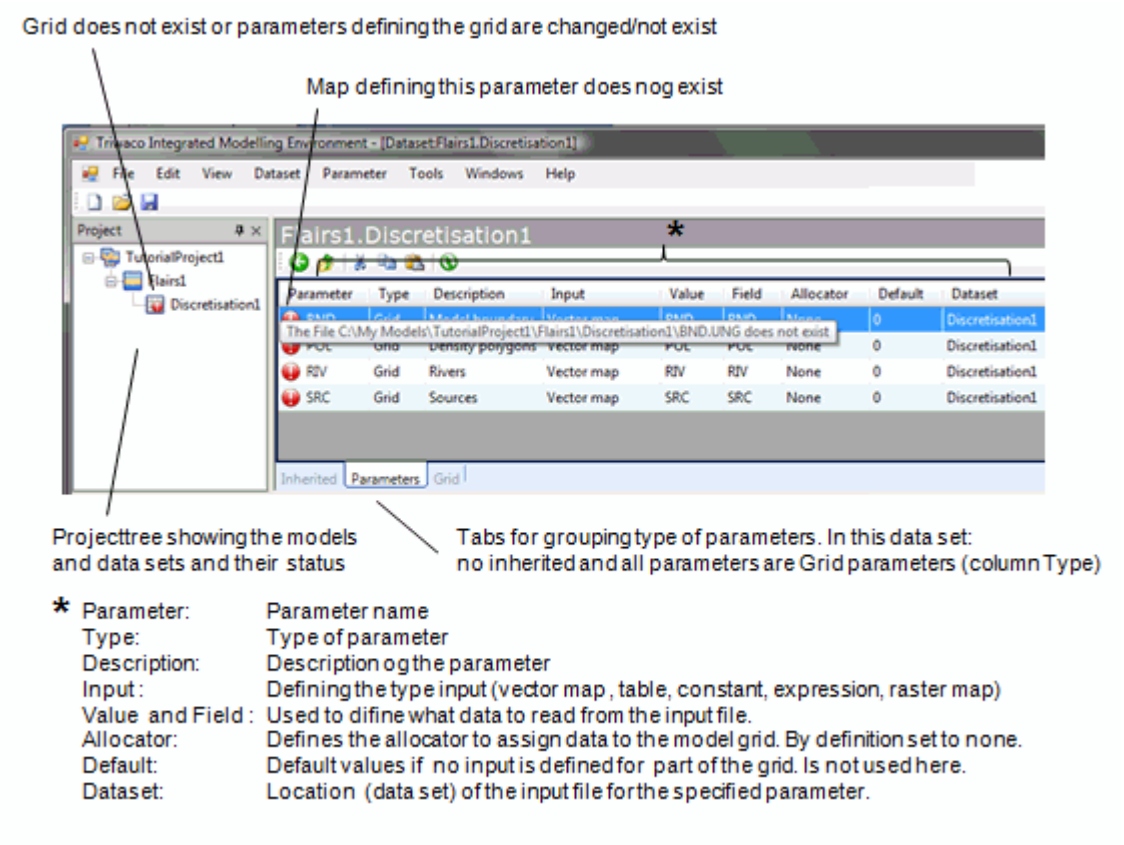
These are options for the programme TESNET that will generate the calculation grid for FLAIRS. The first option defines the EPFIX: Minimal distance between 'Fixed points', e.g. points defined as vertices of the boundary and the rivers or as sources. The value 10 means that when two or more fixed points are closer than 10 meters they are snapped to one fixed point. The second option defines the EPPOL: Minimal distance of points within a density polygon, expressed as fraction of the nominal distance defined for the polygon.



For grid generation and better description of groundwater flow near sources is possible by appending support circles. It is recommend though not necessary when the model is going to be used for pathline calculations. The Support circles allow the user to define a locally very dense grid, which improves the results of the calculation of groundwater flow in the vicinity of abstraction or infiltration wells. Because of the nature of the finite difference grid this option is available for finite elements only. By selecting the appropriate items from the dialogue window under the button Advanced allows the user to add one or more Support circles to the source nodes. The user can choose from a number of predefined radii and sets the number of nodes to be generated on the support circles. The third option in the program options finally defines the number of nodes on the density circle. Since we will use the model to carry out pathline calculations 'Copy' the values from the figure and select 'Next'.

The next window is used to define parameters for the specified dataset. In this case the parameters model boundary, fixed points and lines (stresses) and node density define the calculation grid:
BND = model boundary
POL = density areas (node density / cell size)
RIV = fixed lines, linear surface water (brooks, canals, rivers) or other line elements (faults, etc.)
SRC = fixed points, sources (wells)

Usually there is no need to make changes to this, so select 'Next' and 'Finish'. The dataset is now created and appears as part of the model (if the dataset Discretisation1 isn't visible double click 'Flairs1'). To show the content of the dataset Discretisation1 double click the dataset or right click and 'Open'. The screen shot below explains the information that is provided by Triwaco about the status of the model, datasets and parameters. As well information that defines the parameters. Currently all parameters have a bad status, in our case meaning the parameter maps still have to be defined. The thing to do in the next steps.


Grid does not exist or parameters defining the grid are changed/not exist
Map defining this parameter does nog exist
Triwaco Integrated Modelling Environment - [Dataset:Flairs1.Discretisation1]
File Edit View Dataset Parameter Tools Windows Help
Project
TutorialProject1
Flairs1
Discretisation1
Flairs1.Discretisation1
Parameter Type Description Input Value Field Allocator Default Dataset
The File C:\My Models\TutorialProject1\Flairs1\Discretisation1\BND.UNG does not exist
RIV Grid Rivers Vector map RIV RIV None 0 Discretisation1
SRC Grid Sources Vector map SRC SRC None 0 Discretisation1
Inherited Parameters Grid
Projecttree showing the models and data sets and their status
Tabs for grouping type of parameters. In this data set: no inherited and all parameters are Grid parameters (column Type)
* Parameter: Parameter name
Type: Type of parameter
Description: Description og the parameter
Input: Defining the type input (vector map, table, constant, expression, raster map)
Value and Field: Used to difine what data to read from the input file.
Allocator: Defines the allocator to assign data to the model grid. By definition set to none.
Default: Default values if no input is defined for part of the grid. Is not used here.
Dataset: Location (data set) of the input file for the specified parameter.

6.2.5 Step 2: Defining the model boundary (define a vector map with DIGEDIT)

A boundary or any map (watercourses, sources, etc.) can directly be defined in Triwaco from several file formats (see text box), like a shape file set up in ArcView or ArcGIS, MapInfo or any other GIS software. For the model boundary we will define the model boundary ourself using standard map editor of Triwaco, DIGEDIT. How to define parameters directly from the different file formats will be explained in other steps of creating the calculation grid and model.

OpenGIS in Triwaco
For definition of parameters the modelling environment follows the specifications provided by the Open GIS Consortium (OpenGIS or Open GeoSpatial) using the Open Source Geospatioal Data Abstraction Library (GDAL).

The implementation of GDAL into our software opens the world of all sorts of data file formats that can directly can be read by the modelling environment. It can handle almost all known GIS formats (and the Dutch standards like Aquo, INTWIS and IRIS). The list of supported formats is ever growing, a selection:

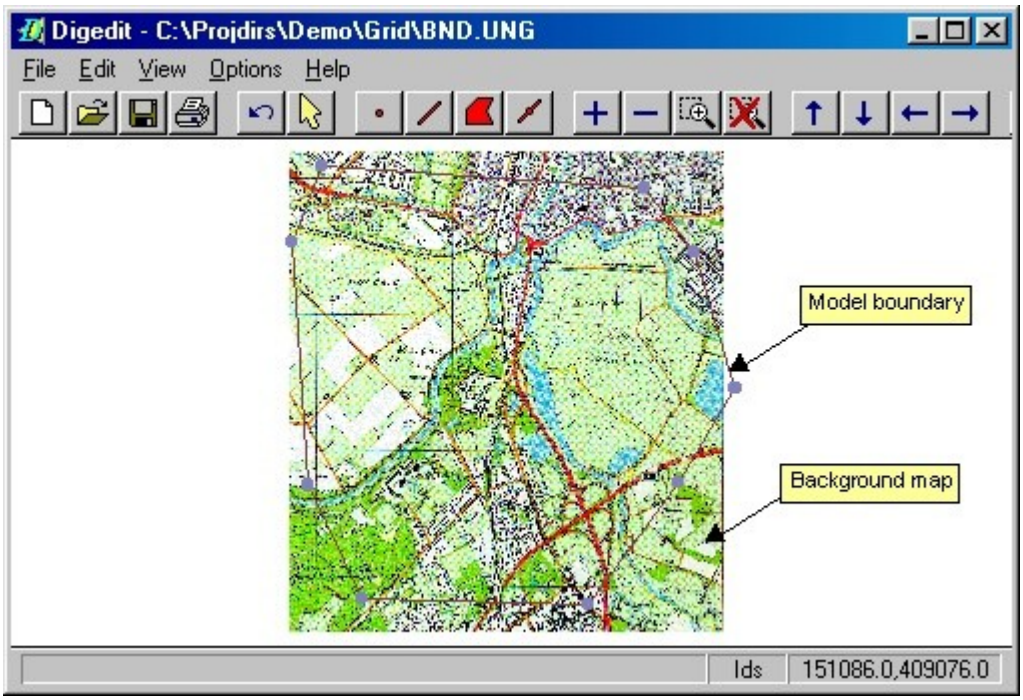
* Raster maps (over 64 formats ; Idrisi, ESRI grids, Erdas, …)
* Vector maps (over 16 format ; ESRI-shape, MapInfo, AutoCAD, …)
* Data bases such as Oracle, MySQL en Access;
* Other well known formats such as Excel, txt en csv.
* Data processing in the modelling environment using expressions and Spatial Queries

Data files in one of these formats can be used as model input without any conversion prior to use in the modelling environment. The modelling environment also supports the conversion of model results to several data file formats

Select 'BND' and open a context menu (Right hand mouse button) select 'Edit'. Even faster is by just double clicking on the dataset to be opened. The default editor will open the map. The type of editor may be changed by the user (see chapter 3), this may be ArcGIS or MapInfo as well. In this case we will use the standard map editor that is provided with Triwaco, DIGEDIT. DIGEDIT is a simple but effective graphical editor that is capable of creating, im/ex-porting Triwaco maps (.ung) or ESRI shape files.
Ignore (click 'Ok') reports as 'Cannot open ..' , this message is generated because no map exists yet.

Now you find yourself in the digitising module (DIGEDIT). It is easy if you can work with a topographical background map for orientation: 'File','Background','Open', and select the background map located My Models\TutorialData\topo.bmp. Topo.bmp is a geo-referenced map.

Draw the boundary by drawing a polygon ( ) Create the model boundary by clicking the corner points in the area; enter the last point (*that doesn't need to be the same as the first point!*) with the right hand mouse; this will close the boundary (polygon); id and value should be 1, name isn't neccesary.



Now save the model boundary (, click 'Ok'). Click refresh (symbool invoegen). Note that the status bullet in the dataset for the boundary parameter is now green. You can check to see if the files are stored in the right directory (the name of which must be the same as the name of the grid dataset). Select and open the context menu for the parameter BND (Right hand mouse button) select 'Explore'. The windows explorer is opened in the directory where the file should be located.

- *It is in DIGEDIT also possible to import a boundary file from a shapfile set up in ArcView or ArcGIS*
- *It is also possible to load a second background map as follows: 'File','Background','Append'.*

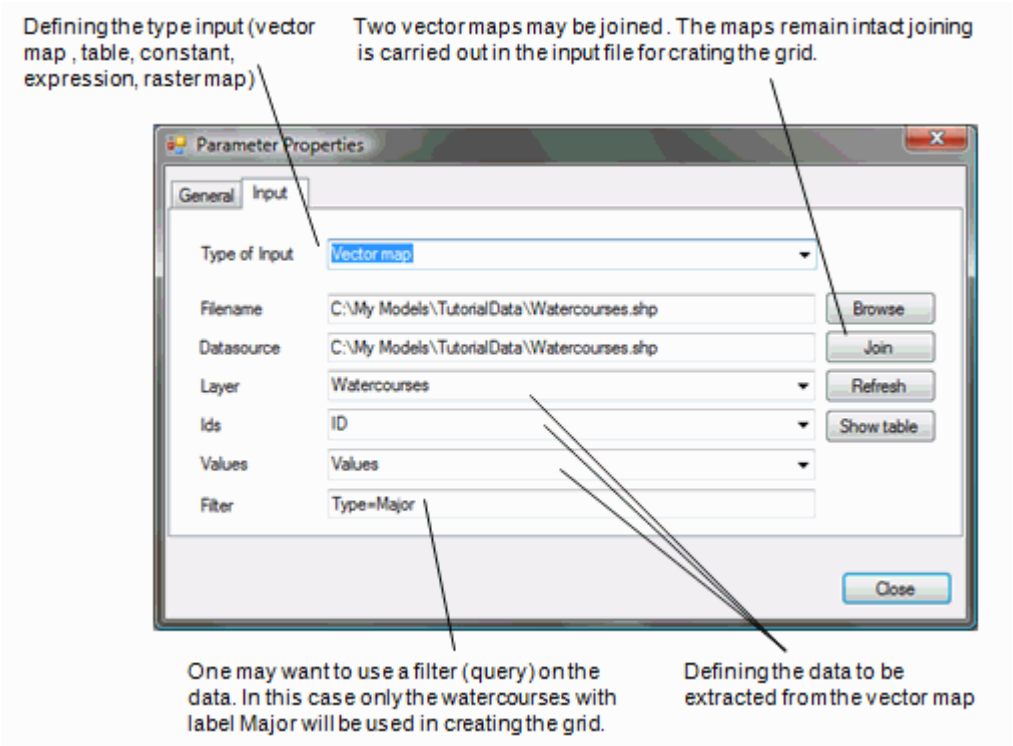
6.2.6 Step 3: Defining the position of watercourses, fixed lines (define a vector map, shape file)

The model boundary was created with the standard map editor DIGEDIT. As mentioned before we can also define parameters directly from several file formats (see text box), like a shape file set up in ArcView or ArcGIS, MapInfo or any other GIS software. In this case we will define the watercourses directly using a shape file which is provided in the directory My Models\TutorialData\Watercourses.shp. There is no need for copying the files in this directory to the project. You could even leave it there for use in a GIS project in the same time.

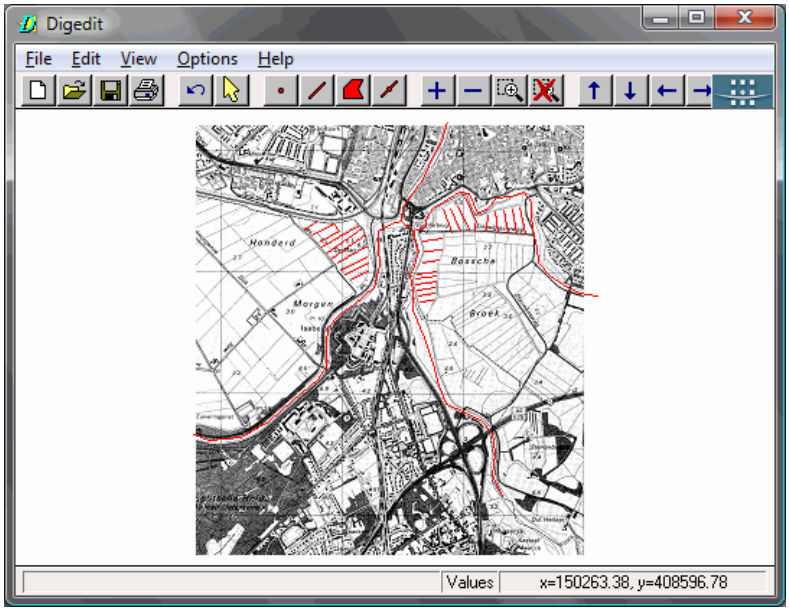
In the dataset select the parameter RIV (which defines the location of watercourses) and on the context menu, select 'Properties'. The parameter properties window has two tabs, General and Input. The General tab gives general information which is also shown in the dataset. For now you can leave this tab as it is. Note that the Status of the parameters says the parameter does not exist.

Now go to the second tab, Input. The figure below shows how to define a vector map. In most cases the parameters that are used to create the grid are vector maps. Use of other type of inputs will be explained when defining model parameters.

Select 'Browse' and locate and select the file watercourses.shp in the directory My Models\TutorialData\. The fields Filename and Datasource will now be the same as shown in the figure. Next thing to do is to define the data that is to be extracted from the vector map. Select the same fields as shown below. Leave the file for filter open for now. And 'Close' the window.



You will now notice that the status bullet of the parameter is green. We will now have a look at the parameter map. Open the file in DIGEDIT as described before (or simply double click). In DIGEDIT mark all check boxes so that all parameter data is loaded. Be shore that ID = ID, Value = Value and Names = Type. You will now see three major watercourses and several smaller ones. The names (type) can be shown by : 'Options','Setup', and check the following check boxes: Hide labels and Names. And close the window. You will now see the same watercourses but with a label. In this model we only want to incorporate the Major watercourses. Instead of deleting the watercourses in the map we leave it intact and will use the filter option. Close DIGEDIT and return to the dataset in Triwaco.



Select the parameter RIV and go to 'Properties'. In the properties window (Input Tab) define the filter as shown in the figure above. With the button Table one can look into the database file of the vector map. Find the column where the labels Major and Minor are defined. You will find that the column is named Type. So to filter the major watercourses the query is as follows: Type='Major' (use single quotation marks for text; use no quotation marks for numbers). So it only will make use of the watercourse with the label Major to create the grid

(watercourses of type 'minor', will remain visible in digedit). 'Close' the properties window.

- *Try to filter this file in another way. Create a filter with ID<4 which should give the same result.*

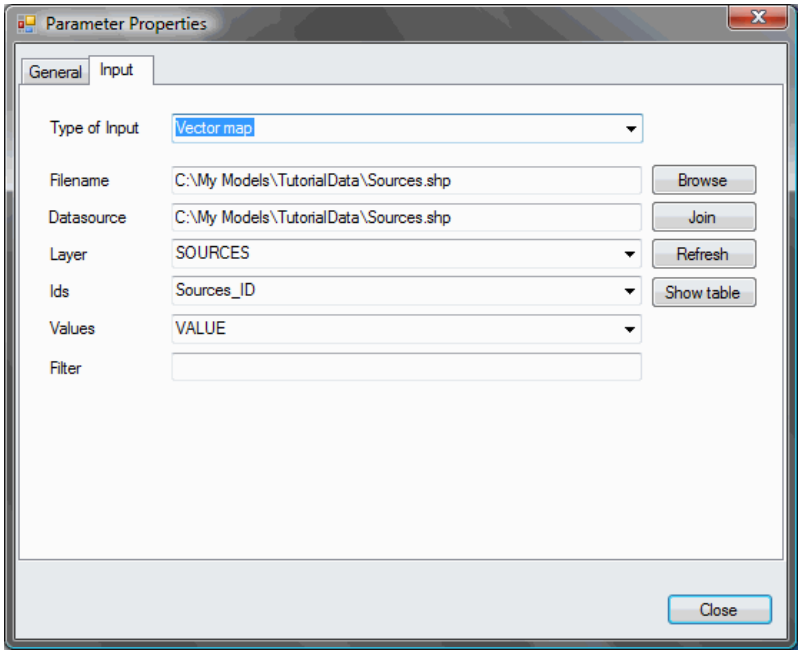
Of course a watercourse map file can be created using DIGEDIT like we did for the model boundary. Draw the water courses by drawing a line ( ) . Now enter the position of the linear surface water (canals, rivers), by clicking the corner and connection points. You are allowed to place line element outside the model boundary. This is recommended if the surface water crosses the boundary. Each line element is closed in the same manner as for the polygon (right hand mouse button). Then DIGEDIT presents you a window in which you can enter the name of the surface water (if you like). Save the data. You can enter as many rivers as you like. Simply repeat the above steps.

- *Delete a water course: Select the river activating the selection mode for line-elements: first* *and then* *,select the water course and press Del on your keyboard.*
- *Select the river activating the selection mode for line-elements: first* *and then* *, drag the watercourse.*
- *Move a point of a water course:* *,* *. Undo: Ctrl-Z or* *or 'Edit', 'Undo'. When no points on the lines are visible select 'Options,'Show Vertices'.*

6.2.7 Step 4: Defining the position of sources, fixed nodes (define a vector map, shape file)

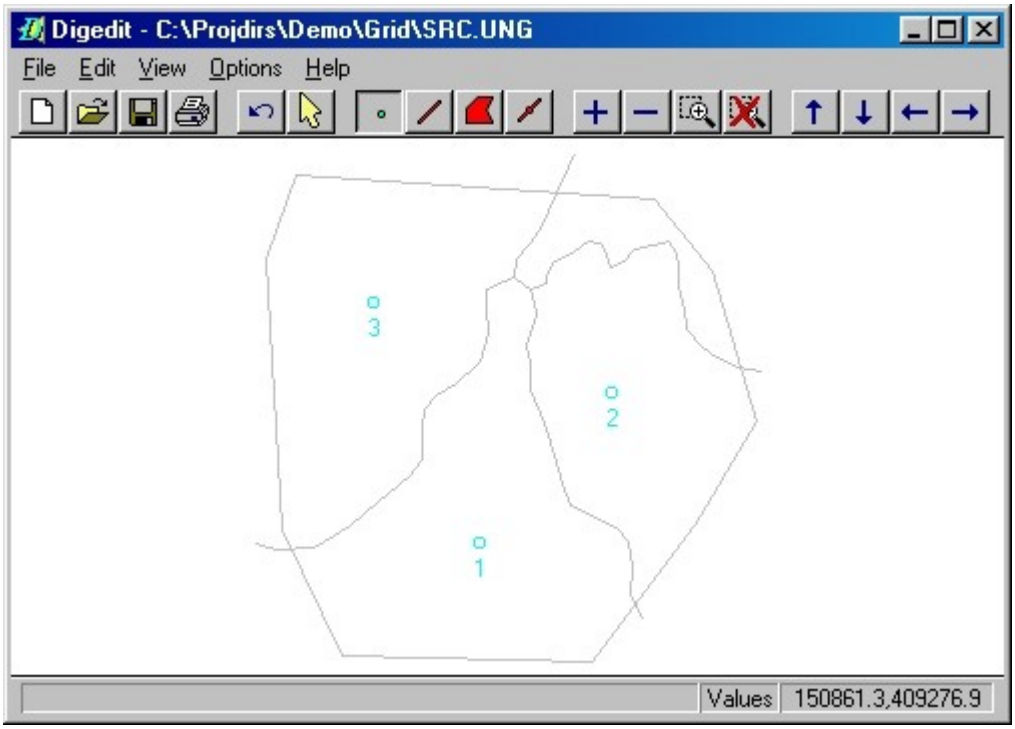
In the same manner as the definition of watercourses we will define the location of the sources. We will define the sources directly using a shape file which is provided in the directory My Models\TutorialData\Sources.shp. There is no need for copying the files in this directory to the project. You could even leave it there for use in a GIS project in the same time.

In the dataset select the parameter SRC (which defines the location of sources) and op the context menu, select 'Properties'. Go to the second tab, Input. Select 'Properties' and locate and select the file sources.shp in the directory My Models\TutorialData\. The fields Filename and Datasource will now be the same as shown in the figure. Next thing to do is to define the data that is to be extracted from the vector map. Select the same fields as shown below. Leave the file for filter open. And close the window.



Of course a source map file can be created using DIGEDIT like we did for the model boundary.

Now enter the position of the sources ( ). Everywhere you click in the area a source is placed. DigEdit will open a window in which you can enter the name (or code) of the source. After which you can carry on an enter a new source. Make sure that the sources are not placed outside the model boundary. You can change the name of sources by: Select the river activating the selection mode for line-elements: first and then , and opening the the context menu for that particular source.



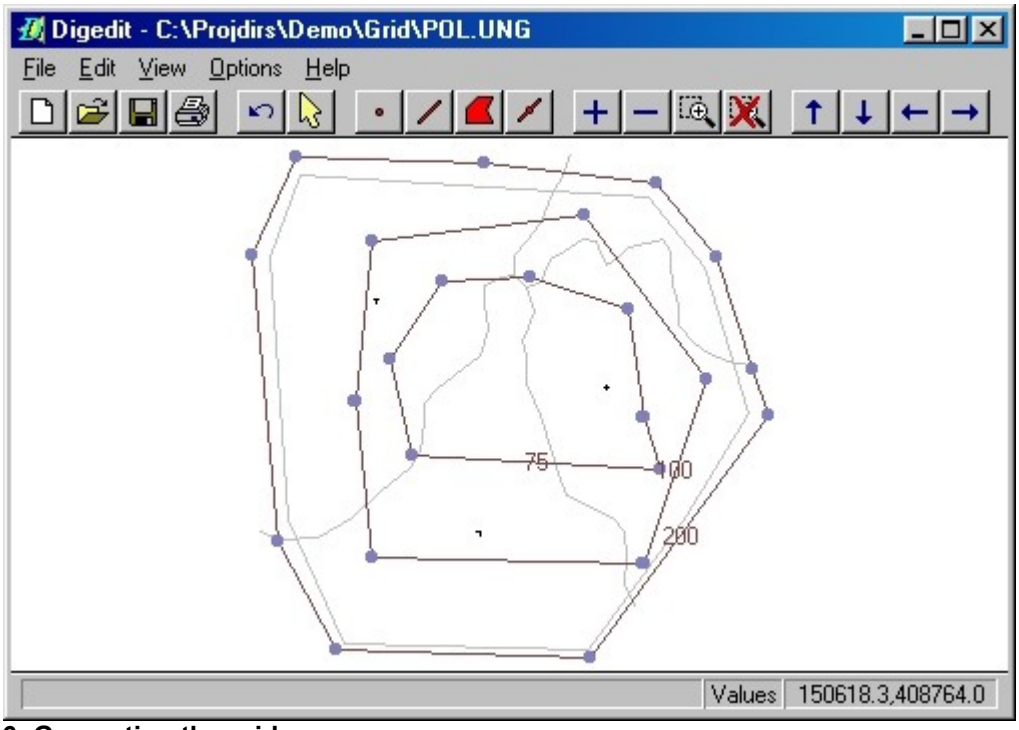
6.2.8 Step 5: Defining the position of node density areas (define a vector map with DIGEDIT)

Since the node density or cell size is model specific it will be created using DIGEDIT like we did for the model boundary. Of course one may also use another GIS editor for that. Open de POL parameter in DIGEDIT. Ignore reports as 'Cannot open ..' . Now load your background map as shown before. Also load (by appending) the map of the model boundary. If you wish, append the maps of the linear surface water and the sources that are stored in the directory My Models\TutorialData\.

Now we will enter the zones with which the node densities are defined for the calculation grid; these are called ‘density areas’. The boundaries of these density areas are entered in the same way as the model boundary by definition of polygons (see Step 2). After closing each boundary, a window pops up with the request to 'Enter ID and value'. Enter the required node distance of the (selected) density area in the field ‘Value’ (usually in meters). Choose the same areas and node densities as shown in the figure.

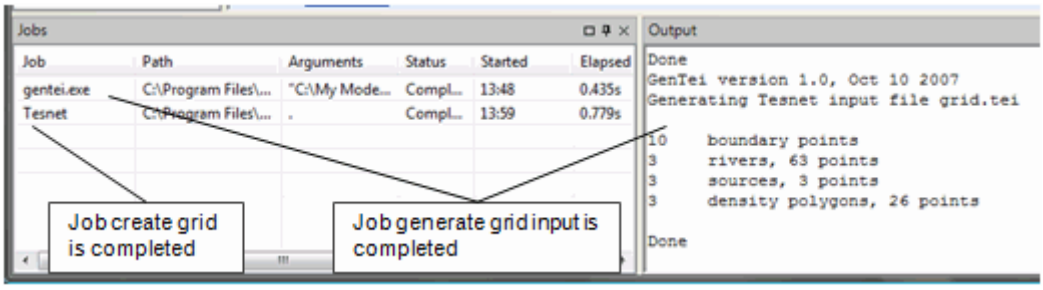
For changing, deleting or moving a density polygon click the icon first, and then (to start editing the polygon) or (if you want to move a vertex). To change node distances: select the polygon by clicking the icon first, and then , and change the value. Deleting polygons can be done by: select the polygon and press Del on the keyboard.

- *The polygon of the last area must be placed completely outside the model boundary.*
- *If you move the corner point of a density polygon where the boundary was closed (i.e. the first corner point entered), you may risk to open the boundary. In that case, delete the line and enter a new polygon.*
- *Adding a vertex to a line/vertex can be accomplished by selecting a line or polygon and then 'Edit', 'Add Vertex' or just by using the Insert key. Then you can indicate the location of the vertex to be added.*



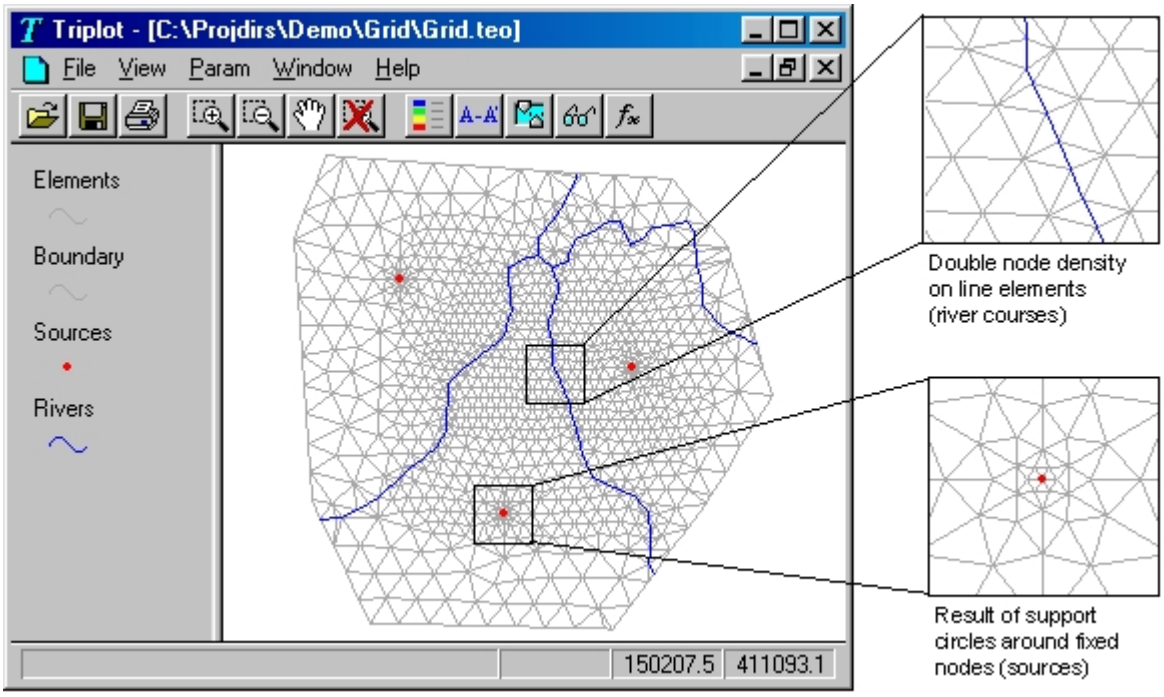
6.2.9 Step 6: Generating the grid

Now all data is entered (all status bullets are green), the grid can be generated. This is done in two steps. First the grid input file is generated after which the grid is created. To generate the grid input file: 'Dataset', 'Generate'. This will create the input file for the grid generator. Triwaco will show in the Jobs pane (if not available do so by 'View', 'Jobs') the progress of generating the grid input file. In the Output pane the log of the grid generator is shown. Note that three rivers are incorporated in the grid. So the filter (step 3) did the job. Also notice three sources and three density polygons were created, as we did in the previous steps. To view the input file: 'Dataset', 'View', 'Input'. This will open the text editor. By default this is Notepad++ an open source editor. But you may define your favourite of course.


Jobs
Job Path Arguments Status Started Elapsed
gentei.exe C:\Program Files\... "C:\My Mode... Compl... 13:48 0.435s
Tesnet C:\Program Files\... . Compl... 13:59 0.779s
Output
Done
GenTei version 1.0, Oct 10 2007
Generating Tesnet input file grid.tei
10 boundary points
3 rivers, 63 points
3 sources, 3 points
3 density polygons, 26 points
Done
Job create grid is completed
Job generate grid input is completed

To create the grid: 'Dataset', 'Run' Again information is provided in the Jobs and Output pane. When an error occurs this is mentioned in the job pane or an error message may appear due to incorrect input. To find out where it went wrong look into the log file. To view the log file: 'Dataset', 'View', 'Print'.
To view the resulting grid: 'Dataset','View', 'Output'.

Now you have entered the presentation module TRIPLOT. You can zoom in and out to check the grid with the well-known icons: . Leave the presentation module and close the grid dataset.



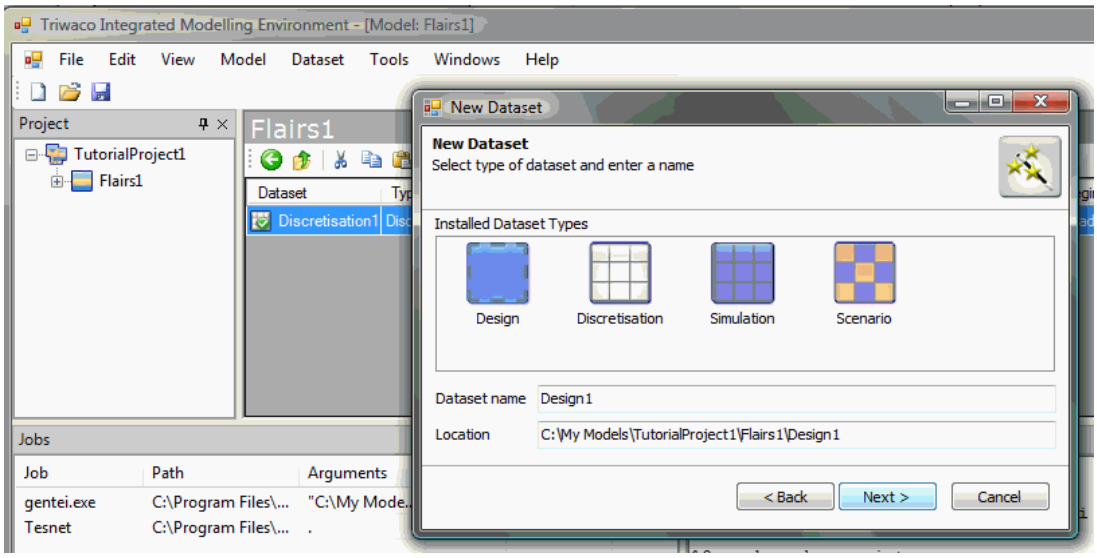
- *Visibility of parameters (like elements, nodes, rivers) as well as the appearance of parameters can be changed in the properties window (context menu or 'View',' Properties' form the pull down menu will open the properties window).*

6.3 Setting up design dataset, the conceptual model set up

The conceptual model is defined in the Design data set. This data set contains the input parameters needed to run the model. The data in this data set is independent from the grid and consists of data like vector maps (ArcGIS, mapinfo), Raster files, excel sheets, etc. The characteristics of each parameter are entered using maps which may contain point values, polygons, lines or constants or a combination. The parameters may also depend on each other using expressions. The default length and time units are meters and days.

6.3.1 Creating a Design data set

Go back a higher level to the level of the model Flairs. This can be achieved by double clicking on it in the project tree or by opening the context menu of the model Flairs and selecting 'Open'.



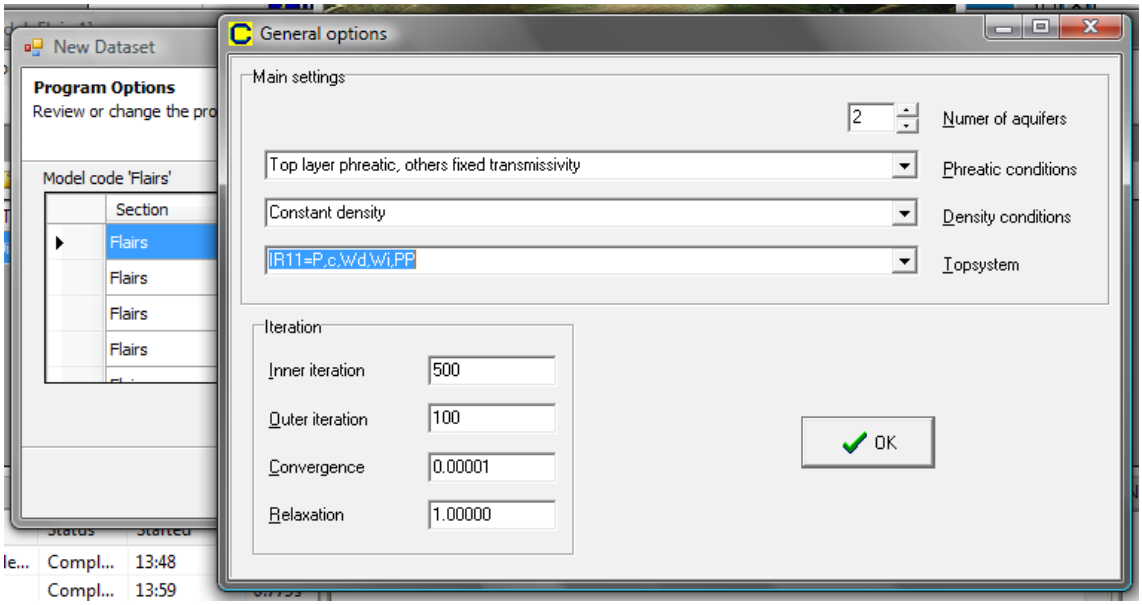
The Design data set is created by: 'Dataset','Add Dataset'. A pup up window will appear the same as when we created the discretisation data set. Again in the first window select next to continue. In the second window one can choose the type of dataset. There are four types of datasets each with its own characteristics and purpose:

- **Discretisation**: Defines the calculation grid (boundaries and stresses like watercourses and wells)
- **Design**: Defines the conceptual model using GIS maps and tables.
- **Simulation**: Here is where the data from the conceptual model is linked to the calculation grid. The model is now prepared to run with the model code.
- **Scenario**: Is similar to the simulation dataset. It is base upon the simulation dataset or another scenario dataset. The dataset is created with parameters linked to the parent dataset. Only parameters that need to be altered for that scenario have to be specified.

Select 'Design'. We will use the de default name so leave as it is, and select next. The following window that appears is for the definition of program options. This window is for more experienced users. We will define the properties of our model via the button 'Advanced'.

In the definition screen you must define data for the model. The choices made here are used to generate the appropriate parameters for the model:

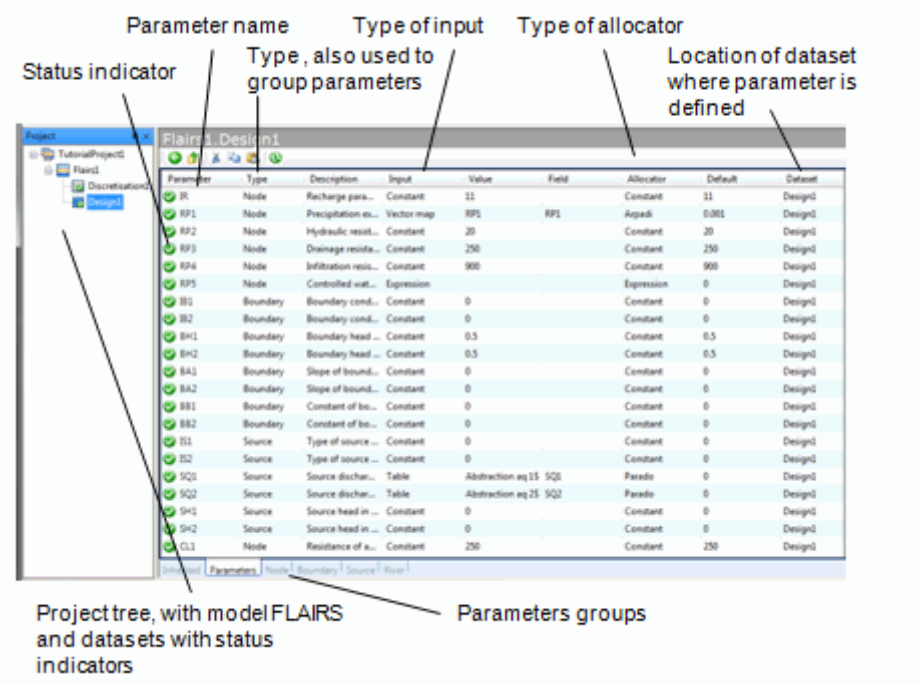
- **number of aquifers** (obvious). The model will contain 2 aquifers so select 2.
- **phreatic conditions**: We want the model to calculate with a variable transmissivity of the first (top) aquifer, depending on the groundwater head in this aquifer. We will use Top layer phreatic (meaning unconfined), others fixed conditions (meaning confined).
- **Density conditions**: this option is used when calculations are carried out whereby the groundwater has a variable density or when used with a sharp interface. In this model no variable density is used.
- **top system number**: type of topsystem (see manual for an explanation of the various Triwaco topsystems). In this case select topsystem no. 11. This type of topsystem is relatively simple and is often used, it contains precipitation excess, drainage and infiltration resistance and a controlled water level.



- *Note for users experienced with MODFLOW. Contrary to most user interfaces in Triwaco the physical features are defined instead of definition of the packages of a model code. When the model input is created this is translated to the appropriate packages of the model code.The advantage is that a model can be set up without hassle of knowing how things are defined in that specific model code.*

The lower part of the general options specifies parameters related to the iteration process of the model code. For now leave this as it is and we will come back on this when the simulation data set is created. Now select 'Next'. The next window summarizes the parameters that will be generated. The amount of parameters and the type of the parameters are based on the settings chosen in the former definition screen. There is no need to make changes here. So select 'Next' and 'Finish'.

The design data set Design1 is now created and filled with the parameters. Except for the topsystem parameter the names of parameters for FLAIRS and several other model codes used in Triwaco are standardized with two characters and one or two values. The characters is the name of the parameter. The numbers designate the model layer. For example SQ4, where SQ stands for abstraction rate and the value 2 shows it concerns aquifer 2.


Parameter name
Type of input
Type of allocator
Status indicator
Type , also used to group parameters
Location of dataset where parameter is defined
Project tree, with model FLAIRS and datasets with status indicators
Parameters groups

The parameters can be divided in 4 types:
- **Node**: parameters covering the whole model area
- **River**: linear surface water parameters or line elements like a horizontal well
- **Source**: source parameters
- **Boundary**: boundary conditions

Every type has a specific way of definition, which is made clear in the next paragraphs.

Within this design data set all parameter information is stored. In other words this data set is your meta database for maps which are independent from the modelling grid. For definition of parameter the modelling environment follows the specifications provided by the Open GIS Consortium (OpenGIS or Open GeoSpatial) using the Open Source Geospatioal Data Abstraction Library (GDAL). The implementation of GDAL into our software opens the world of all sorts of data file formats that can directly be read by the modelling environment. It can handle almost all known GIS formats (and the Dutch standards like Aquo, INTWIS and IRIS). The list of supported formats is ever growing, a selection:

* Raster maps (over 64 formats ; Idrisi, ESRI grids, Erdas, …)
* Vector maps (over 16 format ; ESRI-shape, MapInfo, AutoCAD, …)
* Data bases such as Oracle, MySQL en Access;
* Other well known formats such as Excel, txt en csv.
* Data processing in the modelling environment using expressions and Spatial Queries

Data files in one of these formats can be used as model input without any conversion prior to use in the modelling environment. In the next paragraphs we will show how these different type of file formats are defined.

- *The names of the data sets are used to make directories.*
- *The next time you open the definition screen you can change the data set description.*
- *If you want to change one of the design settings, delete the data set in the list of data sets and start again: Select the data set, 'Data set' , 'Delete'. It is there for recommended not to store your data in the design data set or make a back up copy of it.*

6.3.2 Input of parameters covering the whole model area (type 'node')

The following parameters are node type parameters:

- The topsystem (parameter code **RP*n*.**),
- The resistance of the aquitards (**CL*n*.**)
- The transmissivity of the aquifers (**TX*n*.**) or the permeability (**PX*n*.**),
- The top level of the first aquifer (**RL1**), and the base level of the first aquifer (**TH1**).

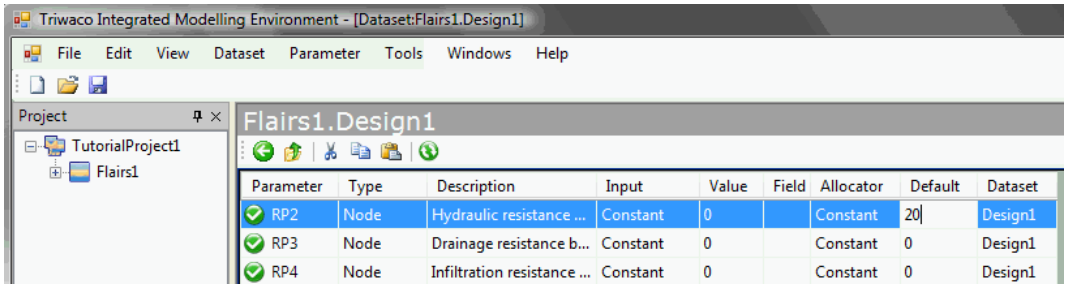
The data to be entered for the different parameters are listed in annex 3b. To help you enter the data four examples are explained in more detail below, illustrating different ways of using file formats to define parameters:

1. **Constant**: Most parameters in our model are defined by a constant (a constant value).
2. **Vector map**: The precipitation excess (**RP1**) is, in this case, entered with a vector map where it is defined by 'polygons' (areas with a constant parameter value) and a default value for areas where no value is defined.
3. **Raster map**: Surface level (**RL1**) defined by a raster map.
4. **Expression**: The average water level in the small surface water (ditches etc.) (**RP5)** that is related to the ground level is defined by an expression.

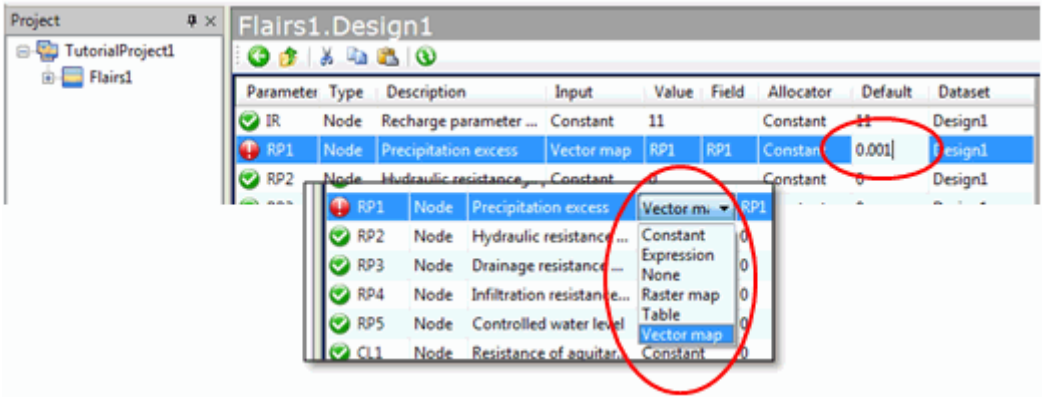
Example 1: Constant used to define parameters with a constant value throughout the model domain

By default the type of input for each parameter is set to constant. To assign a constant value to a parameter is relatively simple to achieve. We will do this for one parameter the values for the other parameters with a constant value are listed in annex 1b.

In the data set Design one can directly change properties of values. The constant value is defined in the column Default. Select the parameter RP2 by clicking once on it. Now click once in the cell with the Default value. You are now able to enter a value. In this case 20 (days) for RP2. Do the same for the other constant values.

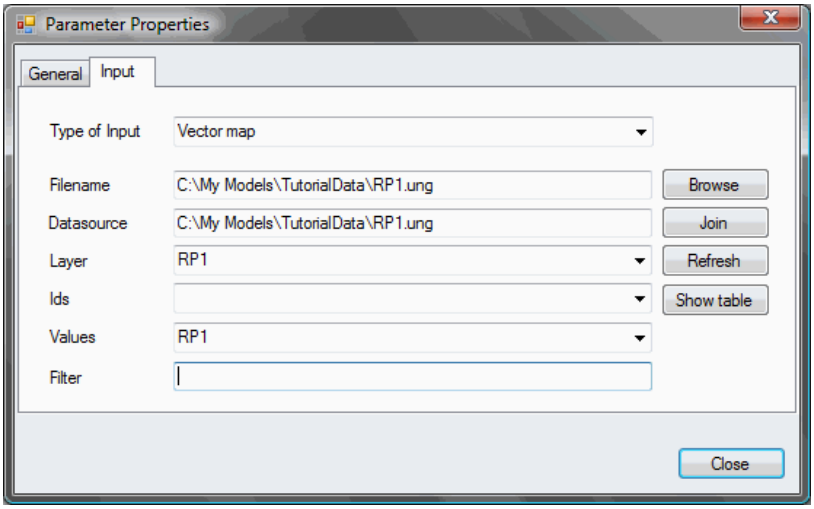

Triwaco Integrated Modelling Environment - [Dataset:Flairs1.Design1]
File Edit View Dataset Parameter Tools Windows Help
Project
TutorialProject1
Flairs1
Flairs1.Design1
Parameter Type Description Input Value Field Allocator Default Dataset
RP2 Node Hydraulic resistance ... Constant 0 Constant 20 Design1
RP3 Node Drainage resistance b... Constant 0 Constant 0 Design1
RP4 Node Infiltration resistance ... Constant 0 Constant 0 Design1

Example 2: Vector map used to define precipitation excess (RP1)
The first thing to do is to set the default value of the parameter RP1 (which is the precipitation excess). As shown in example 1 select the parameter RP1 by clicking once on it. Now click once in the cell with the Default value. You are now able to enter a value. In this case we will use a value of 0.001 m/day (which corresponds to 1mm/day, the average for the Netherlands). Secondly click once in the cell with Input a pull down menu will appear. Select Vector Map. In this way most of the general properties for each parameter can be defined.



Now we will create an Vector Map using DIGEDIT and create a map with polygons (Double Click parameter RP1 to open DIGEDIT). Enter the desired value of the points enclosed by the polygon. Polygons can be changed in the same way as explained for the density areas for the calculation grid. You can check at any moment which part of the area is filled with isoplanes by clicking the item 'Fill polygons' that you find in the sub-menu Options/Setup; the 'white spots' will remain white, the rest is coloured. Save the file RP1 and close DIGEDIT.

If you are not sure, the file used for this tutorial model is enclosed in \TutorialData\RP1.ung and RP1.par. In that case in DIGEDIT choose 'File', 'Append' and select the file. Then save and close DIGEDIT.

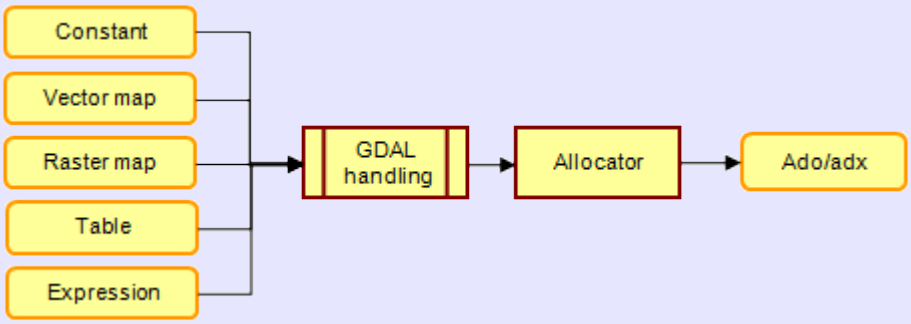


Now you are back in Triwaco in the design data set. The third thing to do is link this parameter to the vector map created. Select the parameter and open the properties window. Select the file in the input tab. In the General tab select the appropriate Allocator. So the question now is what is an allocator used for. An explanation is given in the text box below.

Transformation of input data to model input
Triwaco's traditional strength is the GIS-based model conceptualization. So a model can be set up using maps and tables without link to a calculation grid, as is the case in most other modelling environments. It is not until the detail level of the model is defined by creating the calculation grid that the data has to be translated and converted to values for each cell or node of the calculation grid.

Translating parameter values defined by maps or tables to a calculation grid is carried out by the modelling environment by what we have named “allocation”. Allocation is the spatial or temporal interpolation or up/down scaling. Allocation may be direct assigning values or by interpolation (Kriging, TIN, Inverse Distance, ...) or by using queries, expressions (processing other allocated data) or scripts. Transformation to model parameter input is, usually, in the triwaco file format (.ado or .adox). The reason for using this standard file format is that data can easily be exchanged between models, all tools and processors can access it, automatic calibration tools will always work with any model, etc. Also very important is that all transformed input data can be visualised with the built in viewers.

The transformation or allocation is handled by GDAL (OpenGIS described earlier). The open and modular structure of triwaco allows to develop and implement programs and allocation techniques of third parties or by the modeller him/herself.



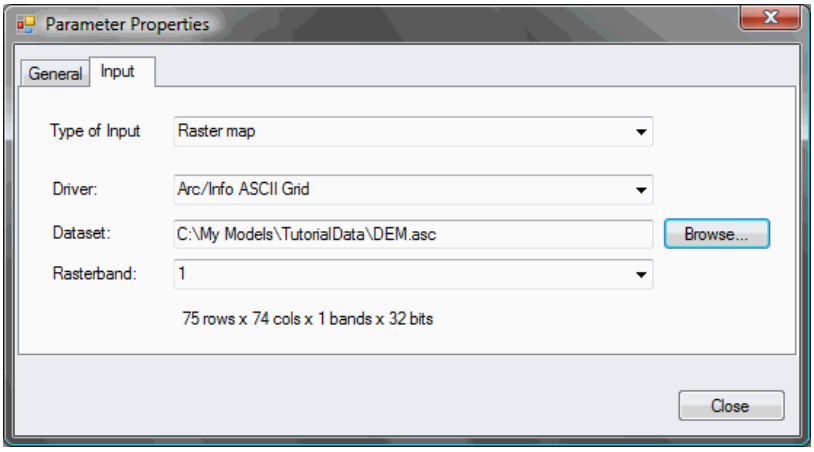
You can choose different allocators depending on the type of input (see annex 1a). In this case, for polygons, select 'Arpadi' (other allocators giving the same result are Warp and Kriging). Close the properties window. You now have defined the first parameter.

- *The polygons may overlap. A grid node that is situated in two planes will be given the parameter value belonging to the polygon with the smallest area (the polygons are sorted by the allocator).*

Example 3: Raster map to define the Surface level (RL1)
In the directory My models\TutorialData\ a digital elevation model is located with the file name DEM.asc. This raster file is in standard ASCII format. However also several other raster files are supported, including ESRI, Idrisi and ERDAS.

In the dataset select the parameter RL1 (which defines surface level, top of aquifer 1) and open the context menu, select 'Properties'. The parameter properties window has two tabs, General and Input. The General tab gives general information which is also shown in the dataset. For now you can leave this tab as it is. Now go to the second tab, Input. The figure below shows the properties Input tab.

Select Type of Input: Raster map. Then select 'Browse' and locate and select the file DEM.asc in the directory My Models\TutorialData\. Triwaco automatically selects the Driver from the OpenGIS database and if there is only one Rasterband it selects Rasterband 1. This can be different if you use satellite imagery. Below this field a summary of the map is given. And close the window.



Now you are back in Triwaco in the design data set. The final thing to do is define the allocator. Select in this case, for a raster map 'Regado'. You now have defined the parameter.

We will now have a look at the parameter map. Open the file in DIGEDIT as described before (or simply double click). You will now see raster map. In this case the map can only be viewed not edited. For editing a raster map other GIS software has to be used. Close DIGEDIT and return to the dataset in Triwaco.

Example 4: Expression to define the topsystem parameter controlled water level (RP5)
A powerful allocator is the Expression allocator. The expression allocator evaluates mathematical expressions between given (allocated) parameters. It can also be used to convert river/node/source parameters to river/node/source parameters. Here we will use a very simple example to define the controlled water level. This is the topsystem parameter **RP5** (controlled waterlevel).

Since an expression allocator uses allocated parameters we will define expressions when we create the simulation dataset. So for now we leave this parameter as it is.

6.3.3 Input of source data (type 'source')

You can choose between sources with a fixed abstraction rate (parameters with code name **SQ*n*.**) or a fixed head (parameters with code name **SH*n*.**). Either one is activated by the value of the parameter **IS*n*.**; the default value is 0 (fixed rate); if you enter value of 1 a fixed head is expected. In our model we use a fixed abstraction rate so there is no need to define the parameter **SH*n***. Since **IS*n*** in this case is 0 (fixed rate) Triwaco will ignore any value defined for **SH*n***.

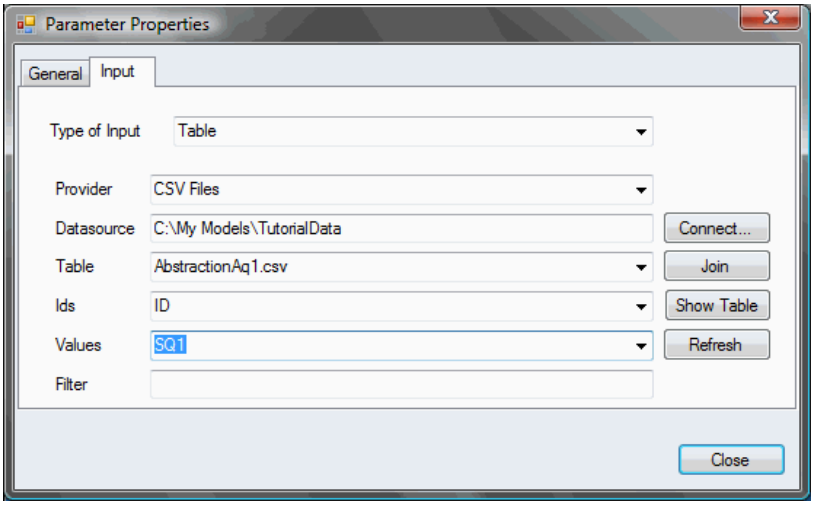
The data to be entered for the different source parameters can be carried out in two ways.

1. **Constant**: Most other parameters are defined by a constant (a constant value).
2. **Table:** The abstraction rate in aquifer 1 (**SQ1**) using a comma delimited file (CSV)
3. **Table**: The abstraction and injection rate in aquifer 2 (**SQ2**) using an excel file (XLS)

Example 1: Constant to define ISn
As explained above in this model we will define an abstraction rate for the second aquifer only. That means that the value for **IS1** and **IS2** (both aquifers) is set to a constant value of 0. The constant value is defined in the column Default. As you may notice the default value is already set to 0 by when the data set was created. So nothing needs to be done.

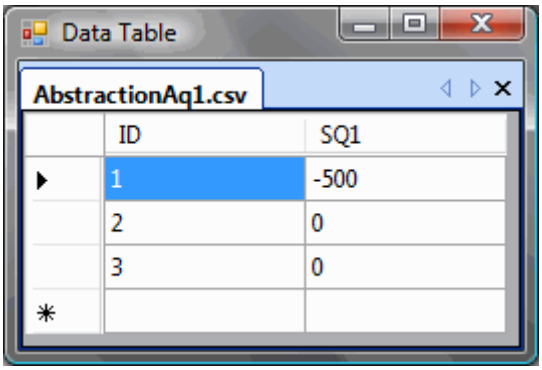
Example 2: Table to define the abstraction rate in aquifer 1 (CSV file):
The only type of input we have not used yet is Table. We will now define the abstraction rate using a table, in this case an comma delimited file. In the directory My models\TutorialData\ the CSV file AbstractionAq1.csv is located. In this case we used an CSV file but many other formats can be used. Before we proceed have a look at the CSV file and note the ID and abstraction rates for aquifer 1 (SQ1) are given. Note that abstraction is negative and rates are in m3/day.

In the dataset select the parameter SQ1 (which defines abstraction rate in aquifer 1) and open the context menu, select 'Properties'. Directly go to the second tab, Input. The figure below shows the properties Input tab.



Select Type of Input: Table. Then in the Provider field select CSV files. Then select 'Connect' and in the next window select 'Browse' and locate and select the file Abstraction.csv in the directory My Models\TutorialData\. You can leave the other fields as they are, they are updated automatically by Triwaco.

Close this window. You are now back in the Properties Window in the tab Input. Select the Table with abstraction rates that is again AbstractionAq1.csv. To check if the correct sheet is selected select 'Show Table'. This comes in very handy since it help to identify the fields ID and Values. Close this window and select the fields as shown in the figure above. Note that a filter can be applied to a table as well, as we saw for a vector map when we defined the watercourse in the discretisation dataset. Close the Properties window.

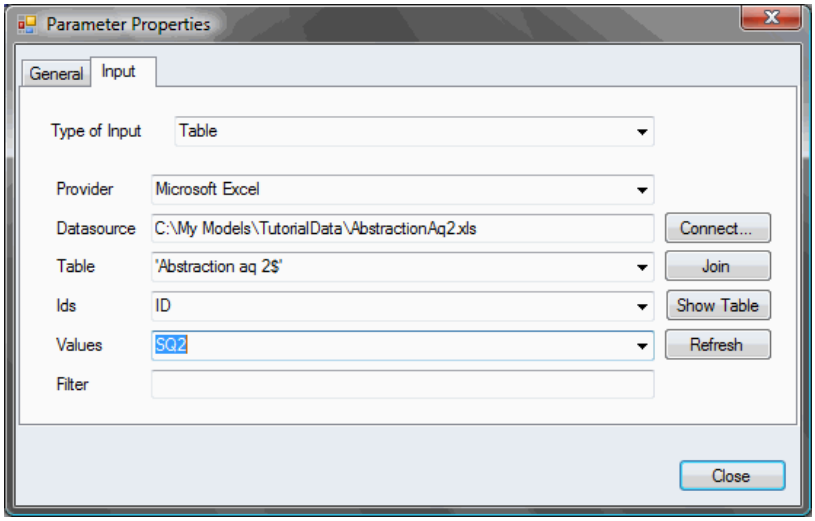


Now you are back in Triwaco in the design data set. The final thing to do is define the allocator. Select in this case, for a source parameter 'Parado'. You now have defined the parameter.

Any changes to values have to be made in the file AbstractionAq1.CSV. If you open the context menu for the parameter SQ1 and select 'Edit' you will find yourself again in the Data Table. You can view the data but not make changes.

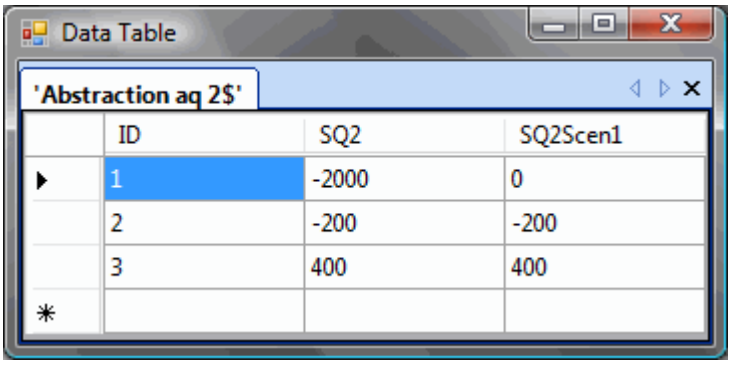
Example 3: Table to define the abstraction rate in aquifer 2 (XLS file):
We will now define the abstraction and injection rate for aquifer 2 also using a table, in this case an excel file. In the directory My models\TutorialData\ the CSV file AbstractionAq2.xls is located. Before we proceed have a look at the XLS file and note there are two columns both with abstraction rates for aquifer 2 (SQ2). The first one is the current abstraction/injection used in this dataset and the second column gives the values which will be used for the Scenario simulation later on. Note that abstraction is negative and injection is a positive value. Rates are in m3/day.

In the dataset select the parameter SQ2 (which defines abstraction rate in aquifer 2) and open the context menu, select 'Properties'. Directly go to the second tab, Input. The figure below shows the properties Input tab.



Select Type of Input: Table. Then in the Provider field select 'Microsoft Excel'. Then select 'Connect' and in the next window select 'Browse' and locate and select the file AbstractionAq2.xls in the directory My Models\TutorialData\. You can leave the other fields as they are, they are updated automatically by Triwaco.

Close this window. You are now back in the Properties Window in the tab Input. Select in the Table with abstraction rates the correct sheet 'Abstraction aq 2'. To check if the correct sheet is selected select 'Show Table'. This comes in very handy since it help to identify the fields ID and Values. Close this window and select the fields as shown in the figure above. Note that a filter can be applied to a table as well, as we saw for a vector map when we defined the watercourse in the discretisation dataset. Close the Properties window.



Now you are back in Triwaco in the design data set. The final thing to do is define the allocator. Select in this case, for a source parameter 'Parado'. You now have defined the parameter.

Any changes to values have to be made in the file AbstractionAq1.XLS. If you open the context menu for the parameter SQ1 and select 'Edit' you will find yourself again in the Data Table. You can view the data but not make changes.

6.3.4 Input of parameters for watercourses (type 'river')

The parameters of the linear surface watercourses (brooks, rivers, canals) are:

1. **RA*n*:** River acivity.
2. **HR*n*:** Water levels
3. **RW*n*:** Width of the water course
4. **CD*n*** and **CI*n*:** Drainage and Infiltration resistance, respectively, for exchange of groundwater and surface water

- *There are several more parameters to be assigned to a line element. These are used for either more advanced options (definition of a river bottom, river clusters and horizontal wells). To learn more about these options look into the next part of this chapter explaining the use of model codes.*

Example: Vector map (water level HR1):
As an example we take the parameter defining the water level of the rivers (**HR1**). Right click 'HR1', then select 'Properties'. We enter the parameter values using a vector map with so called linked points. Select for input 'Vector Map' and for allocator 'Parriv'. Then 'Close' the properties window.
Now open the HR1 map file in DIGEDIT ('Parameter', 'Edit'). Ignore the map does not exist. Next step is adding the surface water map that was previously made for grid generation: 'File', 'Append'. Look for the file 'watercourses.shp' in the directory My models\TutorialData\.

Mind the fact that you did not open this map as a background map this time! Instead you opened it to perform operations on it.

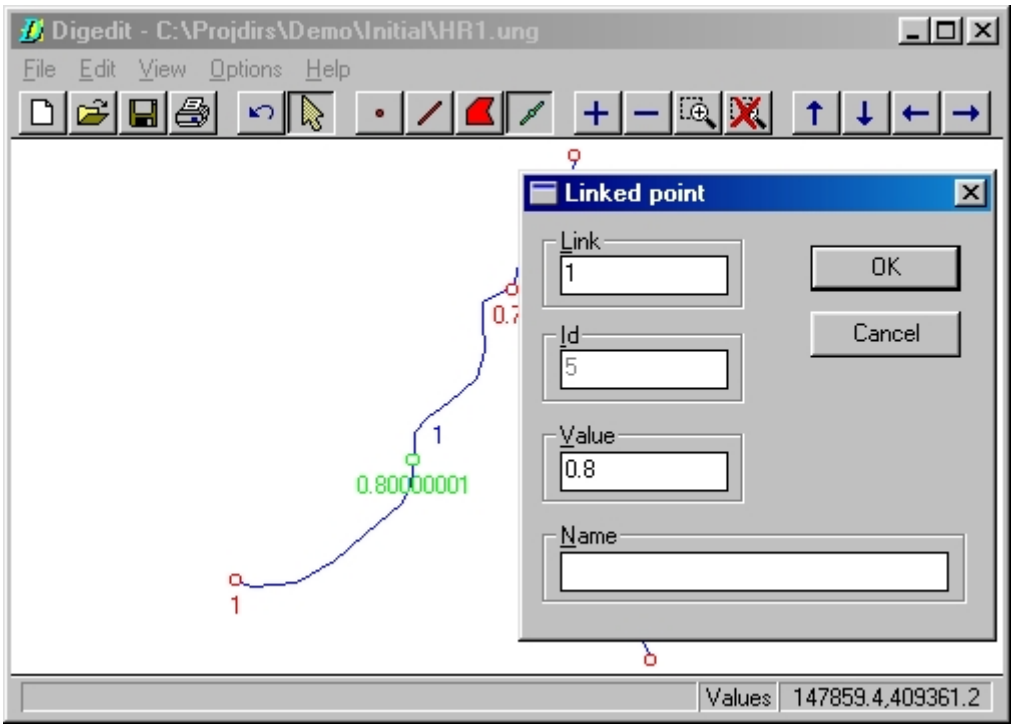
The values of the river parameters are defined by so-called 'linked points'. These linked points are placed on, or within a short distance of, each watercourse. The allocator interpolates between these points, or extrapolates outside these points. Now enter the position of the linked points ( ). For each point you will be asked for the ID of the line to be linked to and a value. Choose as link the ID of the watercourse (that is why we appended the watercourses.shp), and enter the parameter value. The ID of the linked point is not important (see for an explanation on Linked Points the next page).

If the watercourse numbers are not shown on the screen, change it in the menu 'Options', 'Setup', 'Label': Ids (and not: Hide labels).

If you wish to change the value of a linked point: click the icon first, and then (to start editing the linked points).

Now save the file. This is done in the standard Triwaco map file format (.ung) because linked points are stored in a different way. In this case the river number to which the point is linked, the coordinates as well as the parameter value are stored in the ungenerated file format (e.g. HR1.ung).

You are now back in the data set. Define as allocator 'Parriv' (if not already done), this one is specifically developed for assigning linked point values to line elements.



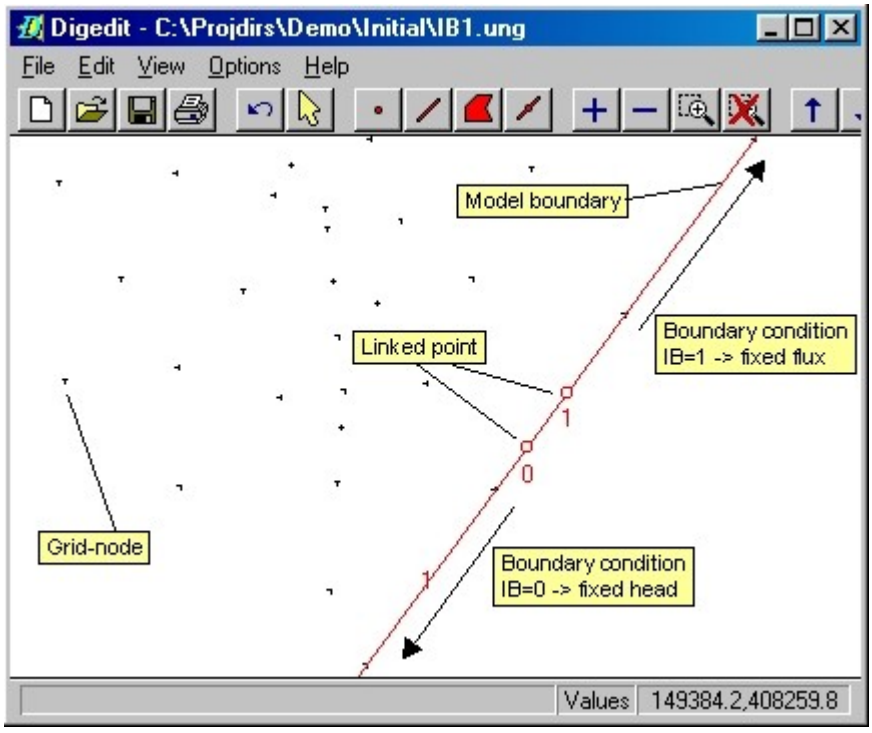
If you did not succeed or don't now what water levels to choose a HR1.ung is prepared and ready to use in the directory My models\TutorialData\. Simply change in the Properties Window the location of the HR1.ung to this one.

- *Each watercourse must be given at least one value (entered by one linked point).*
- *Linked points do not need to be placed exactly on the watercourse; a short distance from it will work too.*
- *If the value changes within a short distance (e.g. over a weir), then you place the linked points at the start and end of this short section.*
- *If you do not enter a value in a map, the default value is taken (defined in the info definition screen).*
- *On a confluence or division point of two watercourses you need to enter values for each of the two. As you cannot stack the input points, you put the points on (or next to) the watercourse, at a short distance of the confluence or division point. If the points are merged, change the Snap distance (Options), delete the points and enter them again.*
- *The River Activity (**RA..**) is a special parameter, that normally has a value (0 (inactive) or 1(active)).*
- *By default the watercourses are active in the first aquifer only; the number in the parameter codes (HR1) marks this. If a watercourse works in a second aquifer too, you must add some parameters:*
  *Choose Parameter/Add/Internal, and look in the parameter list for the parameter type: 'River activity in aquifer', select this type. The parameter is added to the data set with the code name RA. Change this in the info definition screen to **RA2** (the '2' indicates that the parameter is related to the second aquifer).*
  *Do the same for the other parameters: **HR** ('Water levels in rivers in aquifer'); **RW** ('River widths in aquifer'), **CD** ('Drainage resistance of rivers in aquifer') and **CI** ('Infiltration resistance of rivers in aquifer'); add a '2' in all cases. Then you enter the values.*
  *The water level (HR2) may be taken equal to the level in the first aquifer (HR1). In the info definition screen you choose as an allocator 'Expression', and enter the relation in the field 'Expression': HR1 (which means: HR2 = 1.0 * HR1). The other parameters can be treated in the same way, or you can enter new values using a map.*

6.3.5 Input of boundary conditions (type 'boundary')

The parameters for boundary conditions are:

1. **IB*n*:** Type of boundary condition (0 = fixed head, 1 = fixed flux),
2. **BH*n*:** Boundary head, if boundary condition is a fixed head),
3. **BA*n*** and **BB*n*:** Parameters to determine a flux over the boundary; both 0 in case of a flux=0; see the manual for other possibilities,



The boundary conditions are entered with linked points also, similar to the parameters for the linear surface water. In our model the boundary condition is a fixed head, which means that IB*n=0.* Consequently you only need to enter values for the head (parameter BH*n*). The other parameters remain zero. The values are entered similar to those for linear surface water. In between linked points the values are interpolated.
To keep the model simple for boundary conditions we will only enter constant values, so for the boundary head we choose a value 0.5m.

If you want you can of course create a different boundary condition. Open for instance the **IB1** map file in DIGEDIT (select in the Properties Window as input: 'Vector Map', then click 'Close' and open DIGEDIT via 'Parameter', 'Edit'). Ignore the map soes not exist. Next step is adding the model boundary map previously made for grid generation: 'File','Append'. Look for the file 'BND.ung' in the directory ..\Flairs\Discretisation1\. The linked points by definition always are linked to 1, since there is only one boundary. Now follow the same steps as explained for the river parameter **HR1**.

After you finished you are back in Triwaco in the design data set. The final thing to do is define the allocator. Select in this case, for a raster map 'ParBou'. You now have defined the parameter.

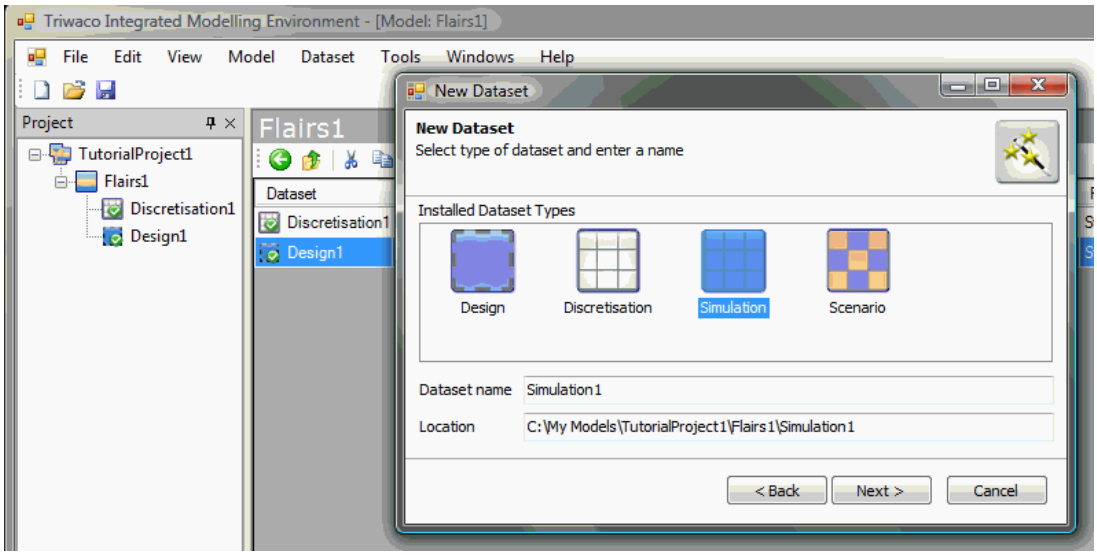
- *If the type of boundary condition (fixed head / flux) should be different at a certain location or stretch of boundary, put two points with different values for IB at both sides of this location (=grid node).*

6.4 Setting up a simulation data set (first simulation)

Up to now the parameter values are stored independently of the calculation grid in tables and maps. The advantage is that the grid can be changed without a change of the original input. The original input is linked (allocated) to the grid in a separate data set, the simulation dataset. This is done only for the conceptual model (design dataset); for scenario calculations the changed basic data (maps and tables) and their allocated values are stored in the corresponding scenario dataset as we will see later on.

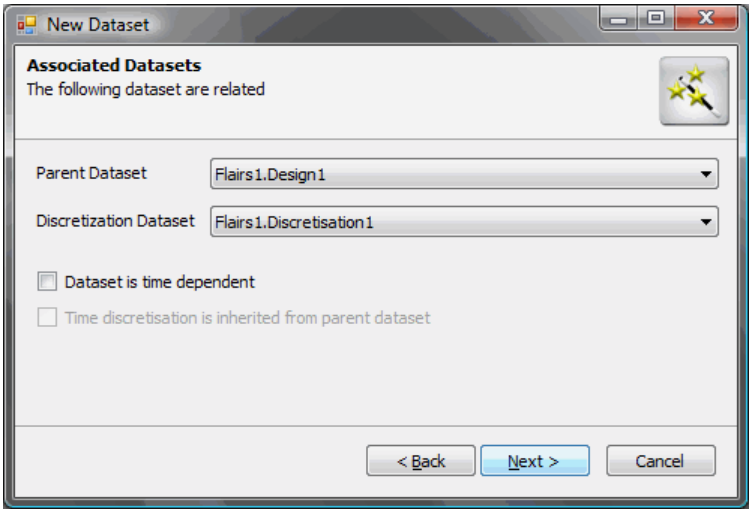
6.4.1 Creating a calibration data set

Go back a higher level to the level of the model Flairs. This can be achieved by double clicking on it in the project tree or by opening the context menu of the model Flairs and selecting 'Open'.



The Simulation data set is created by: 'Dataset','Add Dataset'. A pup up window will appear the same as when we created the discretisation data set. Again in the first window select next to continue. In the second window one can choose the type of dataset. There are four types of datasets each with its own characteristics and purpose:

- **Discretisation**: Defines the calculation grid (boundaries and stresses like watercourses and wells)
- **Design**: Defines the conceptual model using GIS maps and tables.
- **Simulation**: Here is where the data from the conceptual model is linked to the calculation grid. The model is now prepared to run with the modelcode.
- **Scenario**: Is similar to the simulation dataset. It is base upon the simulation dataset or another scenario dataset. The dataset is created with parameters linked to the parent dataset. Only parameters that need to be altered for that scenario have to be specified.

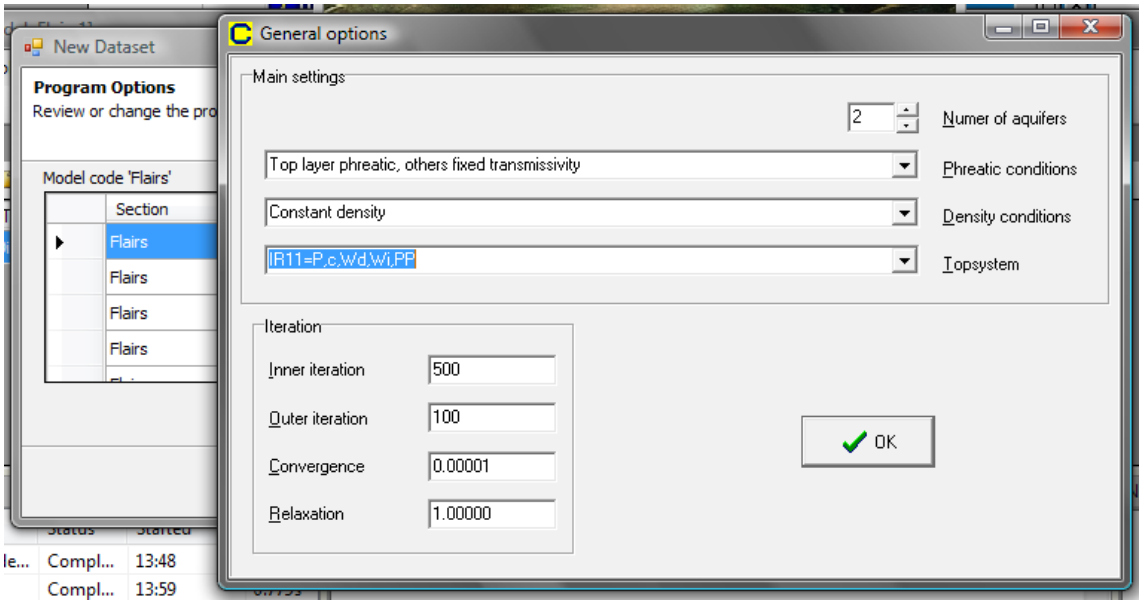


Select Simulation. We will use the de default name (Simulation1) so leave as it is, and select next.

The Simulation set combines the conceptual model (Design dataset) with a calculation grid (Discretisation dataset) to create a model (Simulation dataset) to run with the specified model code. In the following window the conceptual model (Design dataset) is selected as the Parent dataset as well as the calculation grid (Discretisation dataset) as the Discretisation dataset. Note that this means you can easily create an alternative grid and create a new simulation dataset (thus a model) based upon the same conceptual model and visa versa. In the same window one may also specify whether the simulation model should be transient. This will be done later so we leave as it is as shown in the figure.

- *Note that the datasets are defined as [Model name].[Dataset name], so here for the Parent Dataset Flairs1.Design1.*

Select next and find yourself in the same properties window when defining the design dataset. Leave everything as it is. All information was inherited from the Design dataset. Select the 'Advanced' button. We will now focus on the iteration. The lower part of the general options specifies parameters related to the iteration process of the model code.



The user can specify parameters related to the iteration process:

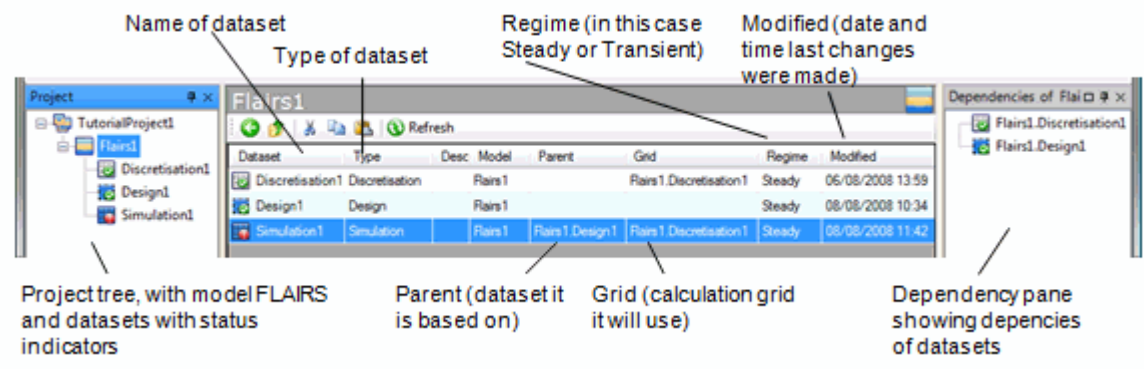
Description	Function
Inner iteration	Sets the maximum number of inner iterations
Outer iteration	Sets the maximum number of outer iterations
Convergence	Sets the criterion for convergence
Relaxation	Sets the relaxation factor

During calculations the model code FLAIRS will pause and display a warning if the maximum number of linear or ***inner iterations*** is exceeded. If the user decides to continue calculations FLAIRS automatically doubles the number of inner iterations. For each ***outer iteration***, the number of inner iterations will be checked.

Calculations will proceed until the number of inner iterations during a single outer iteration equals 2 or less or until the maximum number of outer iterations is reached. Apart from the maximum number of iterations, the user has to specify a ***criterion for convergence***. The program checks whether or not differences are less than the criterion specified. The initial conditions for each outer iteration depend on the head change between (outer) iterations. In case of badly converging systems a ***relaxation factor*** may be defined. In that case the head change between (outer) iterations is multiplied with the relaxation factor. This causes a more stable iteration process but also results in smaller head changes, thus requiring more iterations to reach a solution.

For now leave the default values as they are, usually no changes to these values are needed. So select 'Next'. The next window that appears will show the message that no new parameters were created and that all parameters were inherited from the design dataset. Select 'Next' and 'Finish'.

You are now back in the list of datasets in the model Flairs. Note that the dataset Simulation has a red status indicator. Also notice that it is depending on Design1 dataset (Parent) and Discretisation1 dataset (Grid).


Name of dataset
Type of dataset
Regime (in this case Steady or Transient)
Modified (date and time last changes were made)
Project tree, with model FLAIRS and datasets with status indicators
Parent (dataset it is based on)
Grid (calculation grid it will use)
Dependency pane showing depencies of datasets

Open the dataset Simulation1. You will automatically be directed in the Inherited tab. The parameters are shown in a different font (italic), than what you have seen before. The reason for this is that it is immediately clear that you are dealing with inherited parameters. Also notice that all status indicators are red, which means they either need to be updated or allocation is not carried out yet.

Allocation is translating parameter values defined by maps or tables to a calculation grid. Allocation is carried out by allocators. Allocation is the spatial or temporal interpolation or up/down scaling. Allocation may be done directly by assigning values or be done by interpolation (Kriging, TIN, Inverse Distance, ...) or by using queries, expressions (processing other allocated data) or scripts. Transformation to model parameter input is, usually, in the triwaco file format (.ado or .adx). The reason for using this standard file format is that data can easily be exchanged between models, all tools and processors can access it, automatic calibration tools will always work with any model, etc. Also very important is that all transformed input data can be visualised with the built-in viewers.

We will now allocate the parameter data to the grid. First we will allocate parameters defined by a constant, vector map, raster map or table. Then, we will focus on allocation using an expression.

6.4.2 Allocation of parameters defined by a constant, vector map, raster map or table to the grid

Allocation for constant, tables, vector and raster maps itself is straightforward, since the parameters including the data and allocator were already defined in the design dataset. Allocation is done as follows:

Single parameter
Select the parameter, open the context menu and select 'Allocate'. The allocation process will start. The progress is shown in the Jobs pane and depending on the type of allocator also information is provided in the Output pane. If all went well the status indicator turned green.

Group of parameters
Since it would take a lot of time to allocate each and single parameter you can also select a group of parameters, open the context menu and select Allocate. Again the progress is shown in the Jobs pane and depending on the type of allocator also information is provided in the Output pane. If all went well the status indicators turned green.

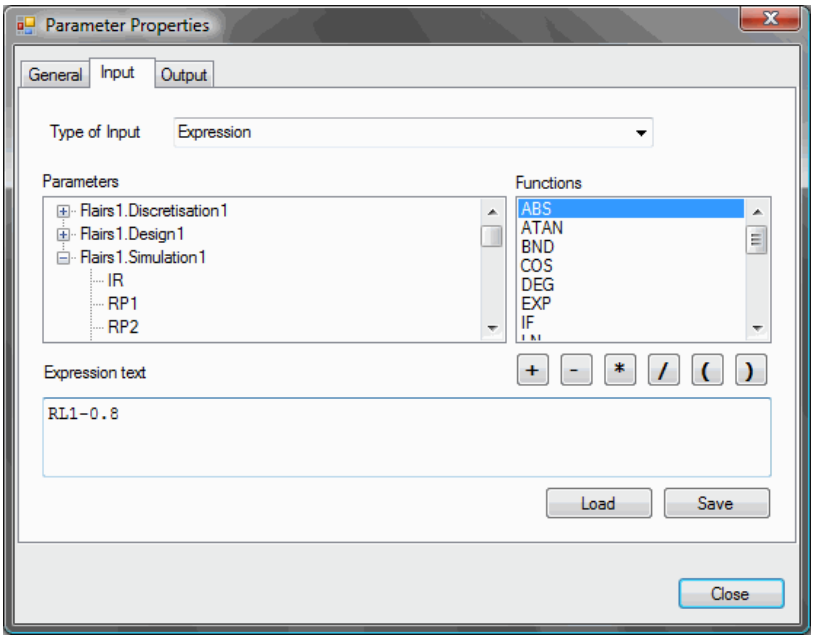
Entire dataset (build)
Shown in next paragraph.

Now allocate all parameters except the controlled water level (RP5) which will be defined and allocated using an expression. If you accidentally allocated this parameter that is not a problem. Simply proceed to the next paragraph. If parameter **HR1** gives the error message “Skipping linked point”, this means a linked point isn't used for the allocation. In this case check the location and links of the linked points in DIGEDIT (open the parameter **HR1** in DIGEDIT). Then allocate parameter **HR1** again.

6.4.3 Definition and allocation of an expression to the grid

When we were defining the data in the design dataset the controlled waterlevel (RP5) was postponed because we want to introduce a powerful allocator, the Expression allocator. The expression allocator evaluates mathematical expressions between given (allocated) parameters. It can also be used to convert river/node/source parameters to river/node/source parameters. Here we will use a very simple example to define the controlled water level. This is the topsystem parameter **RP5** (controlled waterlevel).

In the dataset select the parameter **RP5** and open the context menu, select 'Properties'. The parameter properties window has three tabs, General, Input and Output. Go to the second tab, Input. The figure below shows the properties Input tab.



Choose the type of input 'Expression'. Note later that the allocator is automatically changed to expression as well. In the field 'Expression' enter a relation the relation RL1-0.8. This relation means that the level of the small surface water, modelled with the topsystem, is taken at a level 80 cm below the surface level RL1. Note that in this case it is assumed that the level of the top of the first aquifer is equal to the surface level.

It is done by selecting the parameter RL1 from the list under 'Parameter' and locate it under [Flairs.Simulation1] and double click on RL1. The parameter will now appear in the Expression text box. Now add the rest of the relation.

Expressions can get as complicated as you desire, an example: IF(RL1>TH1,RL1,TH1+0.01) (See also *annex 1a)*. For which the Functions in this input tab can be used to create these expressions.

Close the window. You are now back in the dataset. The only parameter with a red status indicator is now RP5. You can also see that in the column Value the expression is shown. You may change the expression by selecting the parameter, open the context menu. A window similar to input tab is opened. There is one difference there is now a button called Validate. In more complicated models and expressions it is a nice options to test if the expression is defined correctly. Press this button en see whether the expression is correct.

Now you can allocate the parameter. Select the parameter, open the context menu and select Allocate. The allocation process will start. The progress is shown in the Jobs pane and depending on the type of allocator also information is provided in the Output pane. If all went well the status indicator turned green.

- *If parameters are related to other parameters by means of an Expression, the independent parameters must be allocated first and the dependent parameters after that. Triwaco will give a warning in the output pane or when you test the expression.*

6.4.4 Allocating an entire dataset (Build) making all parameters up-to-date

Since modelling is a process of entering data, calibration adapting and changing maps and parameters keeping track of all changes made is difficult. Triwaco already provides a lot of usefull information via the status indicator and dependencies. However with a lot of data you want as quickly as possible allocate all necessary parameters to do another simulation. For this there is the option 'Build'. This option checks the status of all parameters in the dataset and allocates them if necessary. It will also check the dependencies of parameters and will in order of dependencies allocate them. So with one push of the button the entire dataset is up to date.

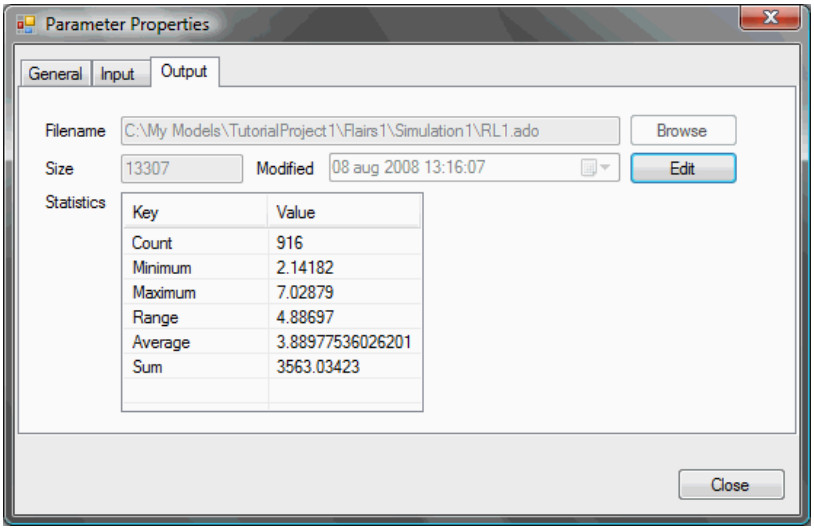
You may test this option even though all parameters at the moment are up to date. From the pull down menu 'Dataset' select 'Build'. The building process starts. In the Jobs pane you will see the progress. In the Output pane information is provided for each parameter. If one of the parameters fails the reason for this is given,so the appropriate action can be taken.

6.4.5 Viewing and checking allocated data (in Triwaco and TRIPLOT)

There is always a chance of error in the model input files (.ado) even though the status indicator may be green. It is therefore recommended to check all the allocated data (adore files) before running a simulation. There are two ways to check the outcome of the allocation. The first one is looking at the summary of the parameter in Triwaco and secondly is using the viewer TRIPLOT.

Checking the allocated data in triwaco

We will check a parameter, in this case RL1, but you may also choose another one. Select the parameter and open the 'Properties Window'. You will now see that next to General and Input tab an additional tab is present, Output. Go to this tab (see figure below for RL1).



The field Filename shows the location of the output file. It is located in the directory Simulation1 and is in the standard triwaco output file format .ado. Also given is the size of the file and the date it was last modified. With Edit the file is opened in the text editor. Below are given the statistics which you can check to see if everything went well, when the parameter was allocated.

Checking and viewing the allocated data in the viewer TRIPLOT

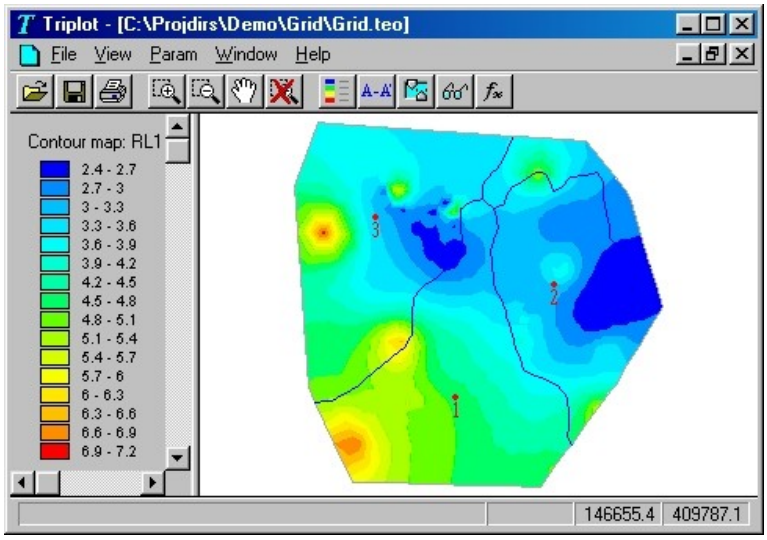
Another way more often used is to use the viewer TRIPLOT. To view a parameter or set of parameters in TRIPLOT, select it or them and then open the context menu and select 'Parameter', 'View'. TRIPLOT is opened the calculation grid is loaded together with the selected parameters. TRIPLOT offers many options of which a few will be dealt with here. For a more elaborate explation on the use and application of TRIPLOT make use of the manual.

You can present the parameter values in several ways:
- Isolines (contours)
- Classes
- Inspector

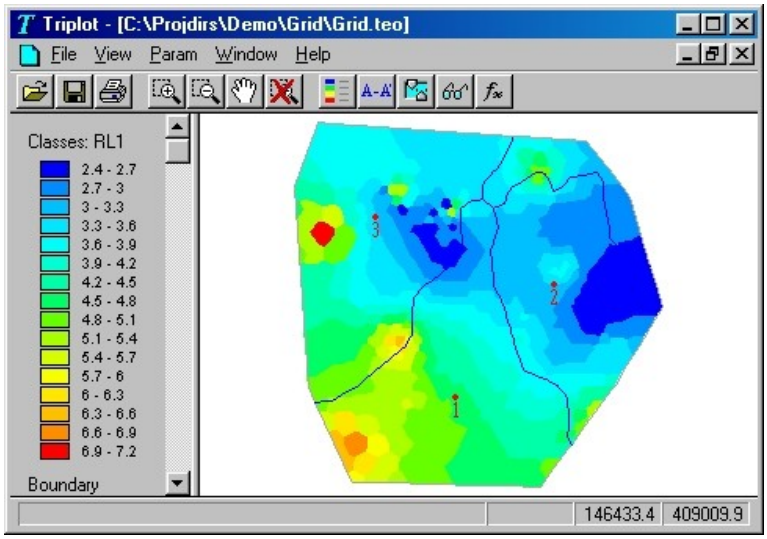
Isolines
In TRIPLOT choose 'Param','Contour', select the parameter. An input screen will appear. The commands in the corner left below (Level) are the most important. A default set of isoline values (levels) has been filled in already, from the minimum to the maximum parameter value.

- Change the total set using Level. You'll get equal differences between the isoline values.
- Change an individual level using Modify; the value and the colour can be changed, and the colour too (click the coloured field to change the colour).
- Change the default series of colours using Colour (command right of Level).
- Insert a level by clicking Insert; Delete will delete a level.
- If you want to save a set of levels, choose Save and enter a file name with the extension 'lvl'.
- Load the saved set of levels using Load.
- Default restores the original set of levels.
- See the manual for the other possibilities of **TRIPLOT**.

If absent add a legend choosing 'View','Legend' and the parameter shown. If you want to change the isolines, you can do so by 'Param','Contour' or 'View','Properties'.



Classes:
The definition of classes is nearly the same as the isoline definition. Choose 'Param','Classify', and do the same as described here before for isolines. If you want to change the classes, you can choose 'Param','Classify' or 'View','Properties'.



- *The numbers in the input table are always the upper limits of the classes. A number equal to a limit is classified into the class with higher values. So, if you enter class limits -1 0 1 2, the classes will be:*

1. [x < -1] ; 2.[-1 <= x < 0] ; 3. [0 <= x < 1] ; 4. [1 <= x < 2].

Inspector:
With the inspector you can point at any location in the model and get a list of values for each parameter loaded.

The inspector is started by selecting . Pointing at various locations causes the program to display coordinates, element and node numbers and values of all loaded parameters for the selected location.

Also useful:
You will frequently use the properties window 'View','Properties' (or click right hand mouse button somewhere in the map). The left part of the window is a list of items shown on the screen; the right part gives a list of all other items that are hidden. You will find - among others - the node values of the chosen parameters ('Node labels').You can also select an item in the list of Visible items, and change the appearance.

Some other options:
Enter a background map in the menu 'View', 'Background map', similar to loading a backgroundmap in DIGEDIT. One may for instance also load a parameter input map to check the allocated parameter. The parameter maps can be found in the Initial data set directory; it's a file consisting of the code name of the parameter and the extension .ung.

- *An extra dimension to the presentation of parameters can be given by the option 'Parameters','Shading'.*
- *Measure lengths and areas by selecting , and pointing the corner points by clicking in the map. The lines will remain on the screen until you zoom in or out or rewrite the screen in another way.*
- *Cross-sections can be made by selecting , and pointing the corner points of the section by clicking in the map. To end a section click the righthand mouse button. Experiment with the options in the section window. Best results are obtained when layer parameters RLn and THn are loaded in TRIPLOT as well.*
- *Try the option of transparency for background map or parameters*
- *All parameters can be exported to vector maps (shape file) or raster maps (.asc).*
- *Visibility of parameters as well as the appearance of parameters can be changed in the properties window (via the context menu or by selecting 'Properties form the pull down menu)*
- *If you have loaded more parameters after each other with the same name, a serial number is added to the code name.*
- *When more than one parameter is contoured/classified you can change the order of appearance similar to a GIS system in the properties window.*

6.4.6 First simulation
When all parameters are allocated to the calculation grid and checked, the model is ready for the first simulation run. The first thing that has to be done is to convert all parameters now in the standard Triwaco file format into modelinput files for the model code as well as creating other input files to run the simulation with the specified modelcode, in this case FLAIRS. (since FLAIRS uses .ado files these files are not converted).

In the Simulation dataset select 'Dataset','Generate', which creates an input file for the model code, based on the parameters defined in the dataset. Selecting 'Dataset' 'View' 'Input' will open the input file created in the text editor.

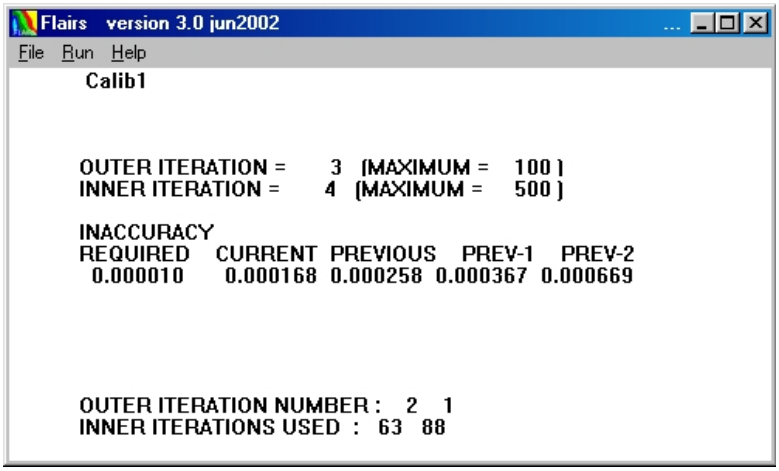
To start the simulation 'Dataset','Run'. A simulation window is opened showing the progress of the simulation. When ready it closes. You can now view the results in TRIPLOT. Select 'Dataset' 'View' 'Output'

When a simulation is finished the following files are produced (located in ..\Simulation1):
- Flairs.flo: result file with piezometric heads and fluxes
- Flairs.flg: log file with a description of the calculation process

- Flairs.flp: print file with water balances for all aquifers, rivers and sources

- *When all input data is defined correctly the model runs without problems. Often however the model will not run because of some errors in the input parameters. There is no standard method for finding the error in the input that causes a problem. Often reading the file FLAIRS.LOG ('Dataset', 'View', 'Log'), where error messages are being displayed, can solve the problem. In the log file it is recorded which parameter causes the problem. Contact the Triwaco Helpdesk if you are stuck.*

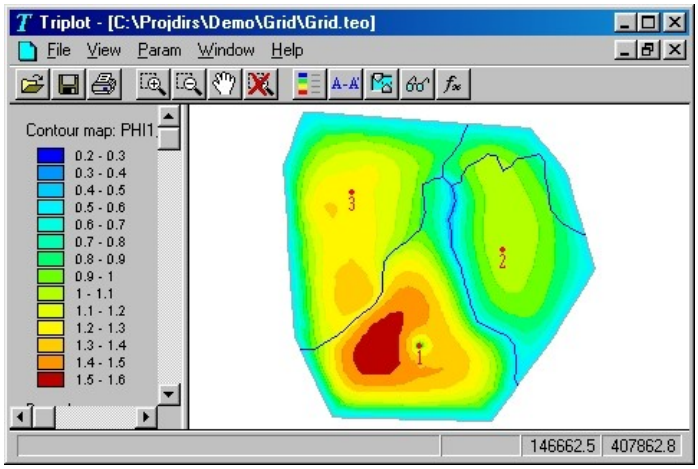
- *For viewing the water balances open the Flairs.flg into a texteditor ('Dataset','View','Print'). The water balances show you how well the conceptual model is and thus the model itself is. For a good model the error in the water balance should be small.*


Flairs version 3.0 jun2002
File Run Help
Calib1
OUTER ITERATION = 3 [MAXIMUM = 100]
INNER ITERATION = 4 [MAXIMUM = 500]
INACCURACY
REQUIRED CURRENT PREVIOUS PREV-1 PREV-2
0.000010 0.000168 0.000258 0.000367 0.000669
OUTER ITERATION NUMBER : 2 1
INNER ITERATIONS USED : 63 88

6.4.7 Viewing and presenting results

When the simulation is finished, you can view the results. Choose 'Dataset','View','output'; again TRIPLOT is opened. The simulation results are automatically loaded, so you can directly select one of the output variables. (If the output file appears not to be selected, Choose 'Param','Load', and select the file 'flairs.flo' in the directory of the simulation data set. Then you can continue with the presentation of the results). Presentation and viewing of the results is carried out in the same way as for viewing allocated data explained earlier. The results consist of the following variables:

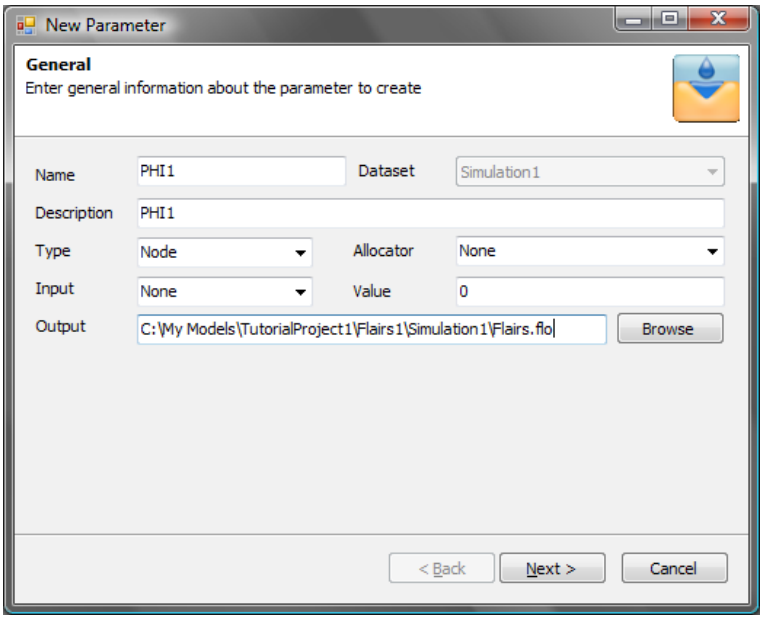
PHIT = head phreatic aquifer (in meter+ reference level)
PHIx = head aquifer x (m+reference level)
QRCH = recharge first (top) aquifer (m/day; positive = downward flux)
QKWx = recharge of aquifer x from aquifer below (m/d; positive = upward flux)
QRIx = exchange flux between groundwater and the linear surface water, in aquifer x (in m^3/day; positive = from surface water to groundwater).



6.4.8 Make results available in the dataset as parameter

Very often the results from the simulation you may want to have available as a parameter so it can be used for post processing or used in expressions.

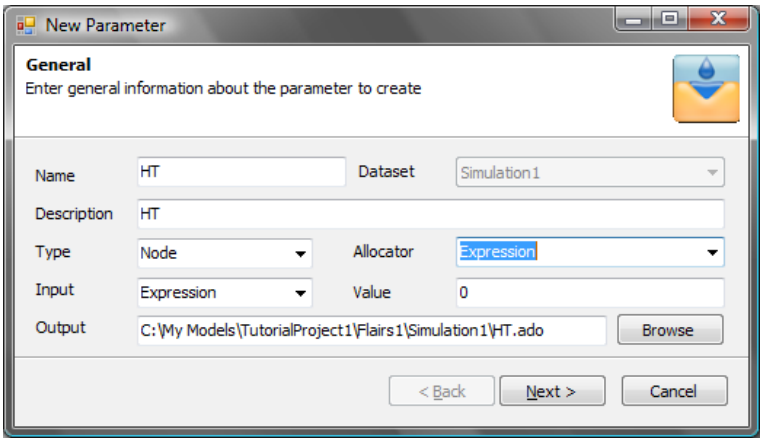
In this case we will show it for PHI1 (calculated head for aquifer 1). Select 'Parameter','New'. In the wizard copy everything from the figure below. Allocator and Input are set to none. The data is extracted from the flairs output file FLAIRS.FLO.



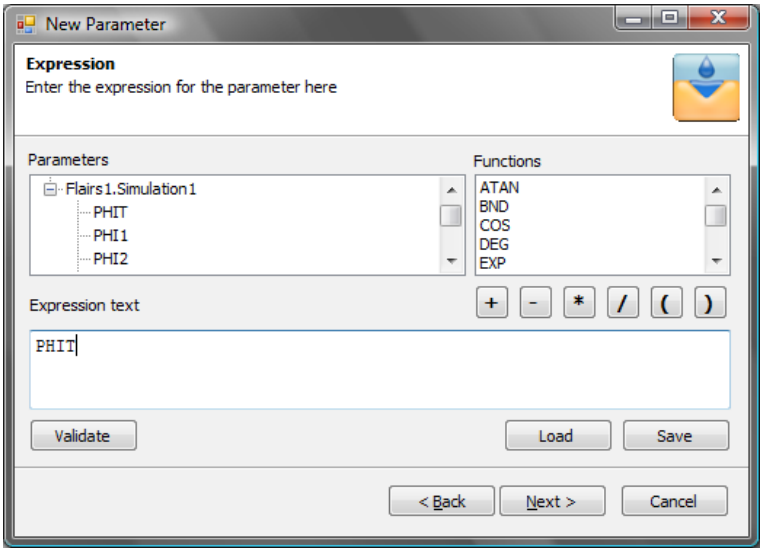
Your first parameter **PHI1** is now created. Now select this file in the dataset and copy and paste it (buttons just above the list of parameters). Select the copied file and change the name and description in the properties window to **PHIT**. Do the same for **PHI2** and the other result parameters fluxes like **QRCH**, **QKWx** and **QRIx**.

6.4.9 Speed up the calculation time of a simulation (extra)

FLAIRS computes groundwater heads/flow by iteration, starting from groundwater heads equal to 0.0. A quicker calculation process may be obtained if you enter initial values for the heads that are closer to the heads to be calculated. After the first calculation with reasonable results, you can enter these calculated heads as initial values. This is done as follows.



Create an other set of parameters: **HT**, **HH1** and **HH2** which are the parameters used to speed up the simulation. Select 'Parameter','New'. In the wizard copy everything from the figure below. Allocator and Input are set to 'Expression'. Then 'Next' and define the expression as we did before and shown below.

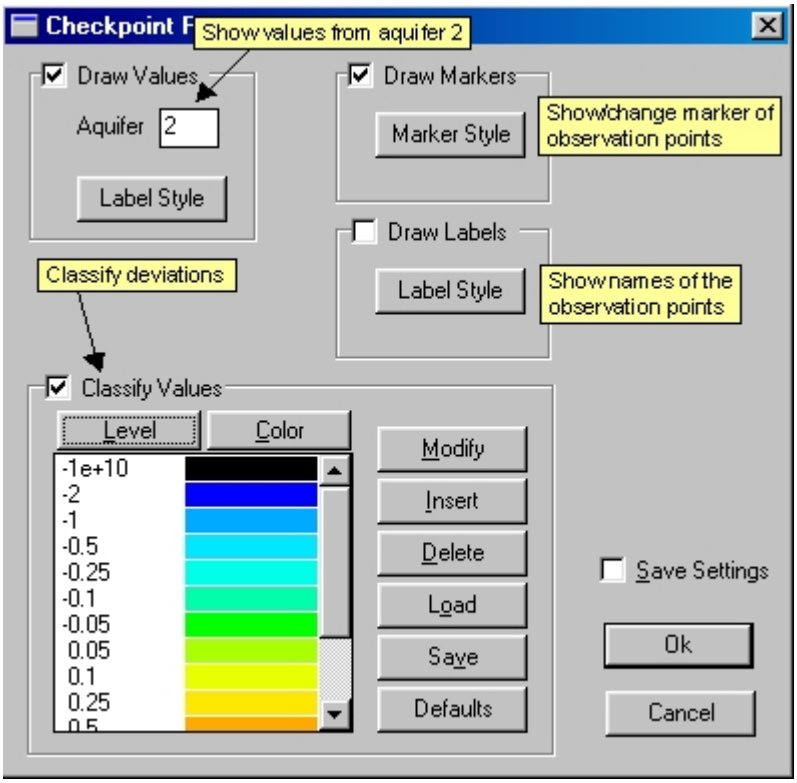


Your first parameter HT was created. Now select this file in the dataset and copy and paste it (buttons just above the list of parameters). Select the copied file and change the name and description in the properties window to **HH1**. Also change the output file name in the Output tab. Do the same for **HH2** and change the expression as well to respectively, **PHI1** for **HH1** and **PHI2** for **HH2**.

We are almost there now. Select the new parameters **HT**, **HH1** and **HH2** and allocate them. Select 'Dataset','Generate', which creates an input file for the model code, based on the parameters defined in the dataset, including the ones we added. To start the simulation 'Dataset','Run'. Note that this will speed up the iteration process.

6.4.10 Comparing simulation results with measurements (calibration)

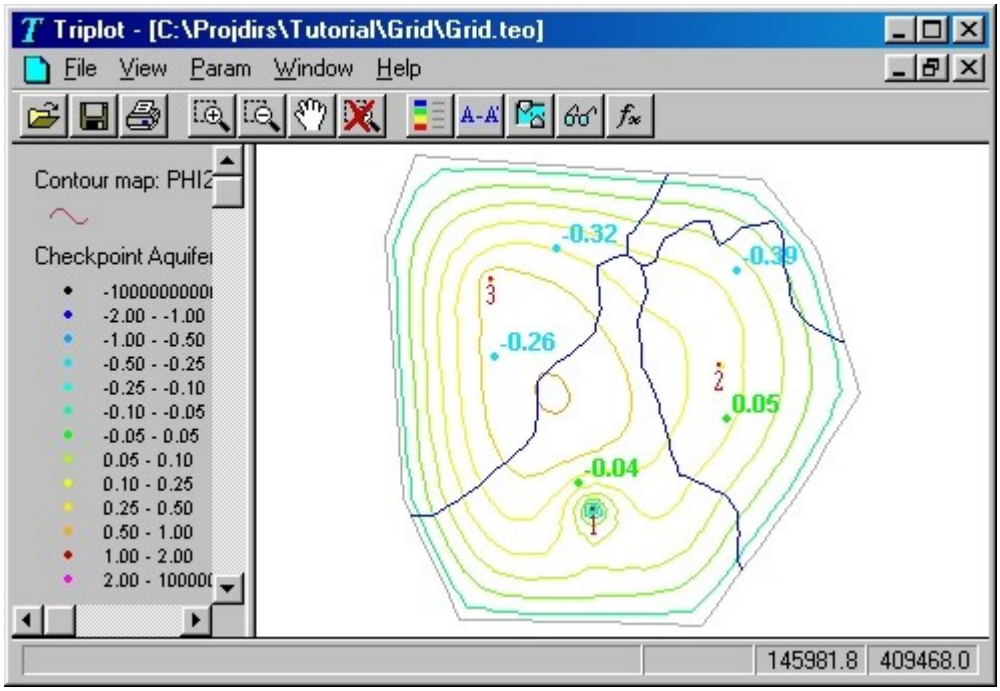
For the calibration of the model you compare the results with measurements. In most cases a model is calibrated using measured groundwater heads, but calibrating using fluxes is also possible. In Triwaco comparison between measured and simulated results is automated.



The only thing you have to do is to create an calibration-input file which has a fixed format. The measured head should be entered in an ASCII-file named **calib.chi**. In annex 2 it is explained how to make calibration input file. In the directory My Models\TutorialData\ a predefined calib.chi can be found. Simply copy this file into the Simulation1 dataset directory. Go back to Triwaco.

After a simulation has run the calculated and measured heads are compared automatically by FLAIRS. The results (stored in the file **calib.cho**) can be viewed in TRIPLOT. Select 'Dataset' 'View' 'Output'. This will load all results into TRIPLOT. In TRIPLOT the calibration results are loaded by 'View',' Checkpoint'. Browse and locate and load the file calib.chi from the Simulation1 dataset directory (this loads the file calib.cho into triplot). Next the properties window is opened. If you like change the settings (e.g. the selected aquifer). The values represent the difference of the calculated value with respect to the measured value. A negative value means that the measured head is higher than the calculated head.

The result should then look something like this.

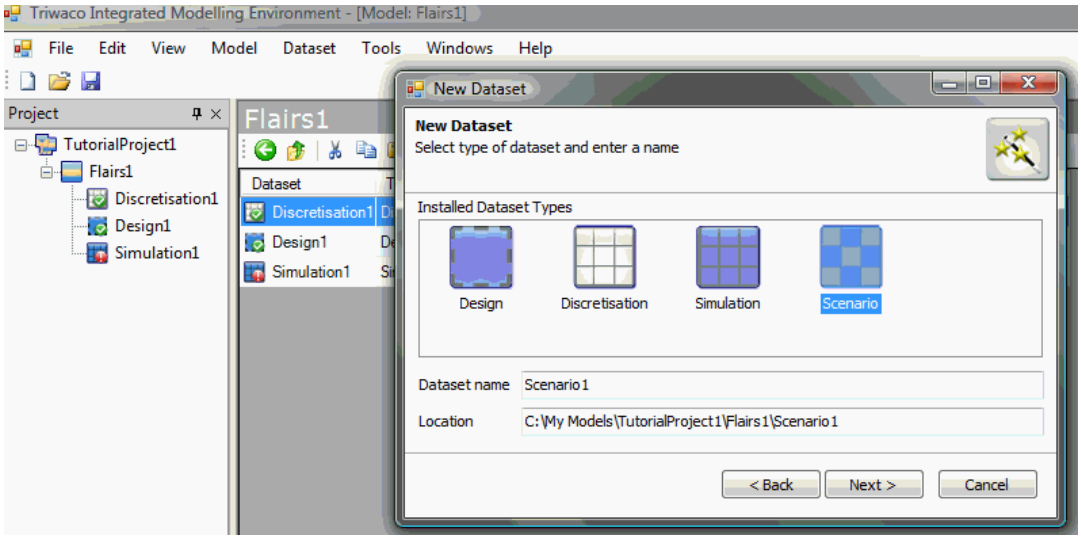


6.5 Setting up a Scenario data set

In most cases a groundwater model or any other model is used to predict consequences of changes made to the water system. These changes usually concern only a few model parameters. Therefore Triwaco has introduced the so called Scenario dataset. For a particular scenario data set the model parameters are inherited from the parent data set (on which it is based) and only the parameters that need to be changed for that scenario have to be defined.

6.5.1 Creating a scenario dataset

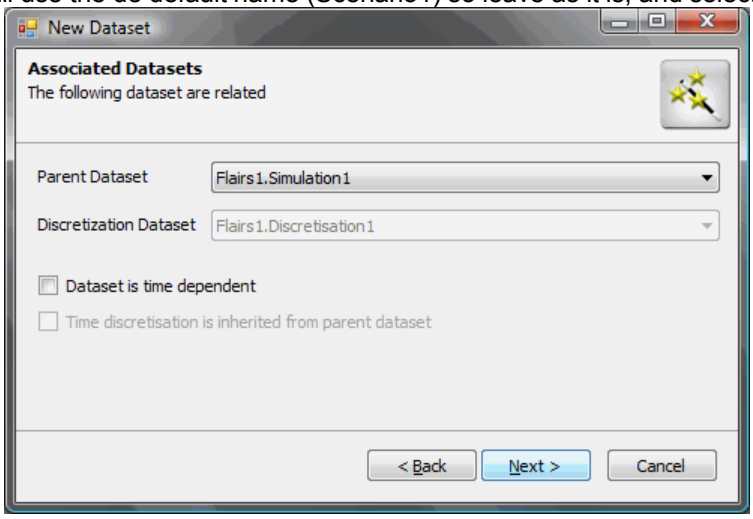
Go back a higher level to the level of the model Flairs. This can be achieved by double clicking on it in the project tree or by opening the context menu of the model Flairs and selecting 'Open'.



The Scenario data set is created by: 'Dataset','Add Dataset'. A pup up window will appear the same as when we created the other data sets. Again in the first window select next to continue. In the second window one can choose the type of dataset. There are four types of datasets each with its own characteristics and purpose:

- **Discretisation**: Defines the calculation grid (boundaries and stresses like watercourses and wells)
- **Design**: Defines the conceptual model using GIS maps and tables.
- **Simulation**: Here is where the data from the conceptual model is linked to the calculation grid. The model is now prepared to run with the model code.
- **Scenario**: Is similar to the simulation dataset. It is base upon the simulation dataset or another scenario dataset. The dataset is created with parameters linked to the parent dataset. Only parameters that need to be altered for that scenario have to be specified.

Select Scenario. We will use the de default name (Scenario1) so leave as it is, and select next.



The Scenario dataset is based upon a Simulation dataset or other Scenario. In the following window that appears select Simulation1 as the Parent dataset. Note that this means you can easily create several scenarios without copying entire directories. This strongly enhances the reproducibility of you modelled results.

In the same window one may also specify whether the Scenario model should be transient. This will be done later so we leave as it is as shown in the figure (previous page). Also (in case the Scenario concerns a transient data set) you can inherit the time discretisation or create a new one.

- *Note that the datasets are defined as [Model name].[Dataset name], so here for the Parent Dataset Flairs1.Simulation1.*

Select next and find yourself in the same properties window when defining the design and simulation dataset. Leave everything as it is. All information was inherited from the Simulation dataset. Select next and finish.

6.5.2 Create a scenario by modifying a parameter

You are now back in the list of datasets in the model Flairs. Note that the dataset Scenario has a red status indicator. Also notice that it is depending on Simulation1 dataset (Parent) and Discretisation1 dataset (Grid).

Open the dataset Scenario1. You will automatically will be directed in the Inherited tab. This is because in fact for the Scenario dataset no parameters are defined, all parameters at this moment would be used from the Simulation dataset. So if you would run a simulation now the outcome is exactly the same as for the Simulation1 dataset (the Parent). Notice that all status indicators are green which means they are up-to-date. Which is no surprise since the dataset Simulation1 is up-to-date.

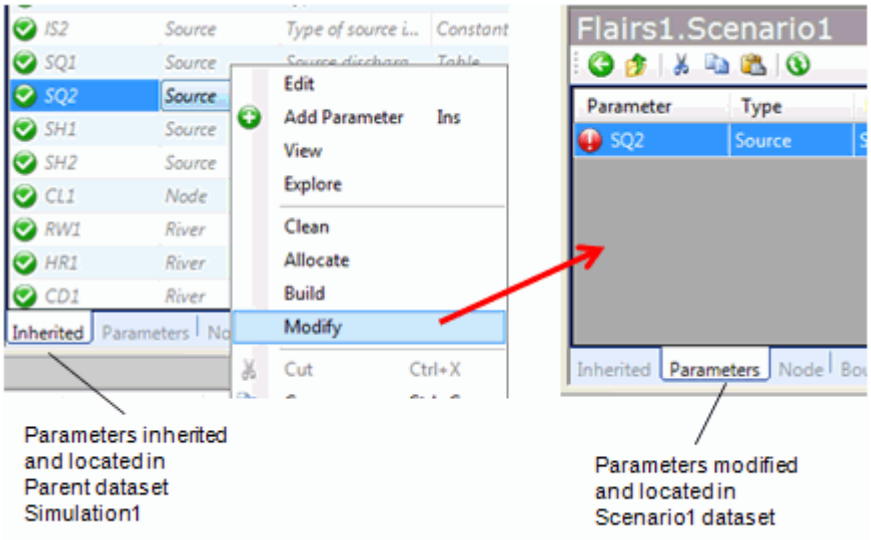
- *Note that all inherited parameters are in italic and all modified parameters are in normal font.*

In the Scenario1 dataset there is a tab Parameters, however it is empty. All the parameters of the Parent data are located in the inherited tab. In the Parameters tab only parameters that are modified with respect to the Parent data set are stored.

E.g.: if you want to calculate the effect of a different abstraction rate in a source, you only have to modify this parameter which then will be moved to the Parameter tab. This is what we will do in this case; we will stop the abstraction from well with ID 1.

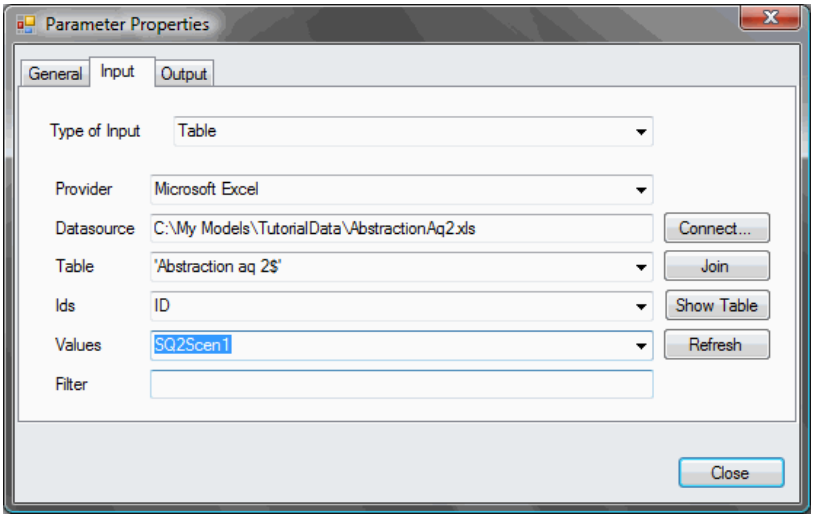
First thing to do is to move parameter **SQ2** (abstraction rate in aquifer 2) from the inherited tab to the Parameter tab; all other parameters remain the same as in the Parent (Siumulation1) data set. This is done as follows.

Open the Scenario data set. Select the parameter(s) that is to be modified with respect to the Simulation dataset set by choosing 'Modify' from the context menu. Now the parameter is moved to the Parameters tab.



Next we need to define the abstraction parameter **SQ2** again using a table, in this case the scenario is already prepared in the existing xls file AbstractionAq2.xls. Before we proceed have a look at the XLS file and note that in the third column (SQ2Scen1) the abstraction rate of well with ID 1 is set to zero (previously -2000).

In the dataset select the parameter **SQ2** (which defines abstraction rate in aquifer 2) and open the context menu, select 'Properties'. Directly go to the second tab, Input. The figure below shows the properties Input tab.



Select Type of Input: Table. Then in the Provider field select XLSfiles. Then select 'Connect' and in the next window select 'Browse' and locate and select the file AbstractionAq2.xls in the directory My Models\TutorialData\. You can leave the other fields as they are, they are updated automatically by Triwaco.

Close this window. You are now back in the Properties Window in the tab Input. Select in the Table the sheet 'Abstraction aq 2'. To check if the correct sheet is selected select 'Show Table'. This comes in very handy since it help to identify the fields ID and Values. Close this window and select the fields as shown in the figure above. Close the Properties window.

Now you are back in Triwaco in the Scenario1 data set. The allocator still is Parado, so the only thing to do now is to allocate the parameter.

6.5.3 Run the Scenario simulation

In the dataset Scenario1 select 'Dataset' 'Explore'. This will open the windows explorer in the dataset directory. Notice that only the modified parameter **SQ2** is physically present in the Scenario data set directory. In the input file for the model code all other parameters will be referenced to the Simulation1 dataset.

To run the simulation choose 'Dataset','Generate' (creates an model code input file, based on the modified parameters in the set and the parameters referenced to the Simulation1 dataset) and then 'Dataset','Run'. A simulation window is opened showing the progress of the simulation. When ready it closes. You can now view the results with 'Dataset','View','Output'.

For a little help on how to modify parameters, presenting the results and in an efficient way combining and processing of data proceed to the next paragraph.

6.5.4 Combining and processing of model output and parameters

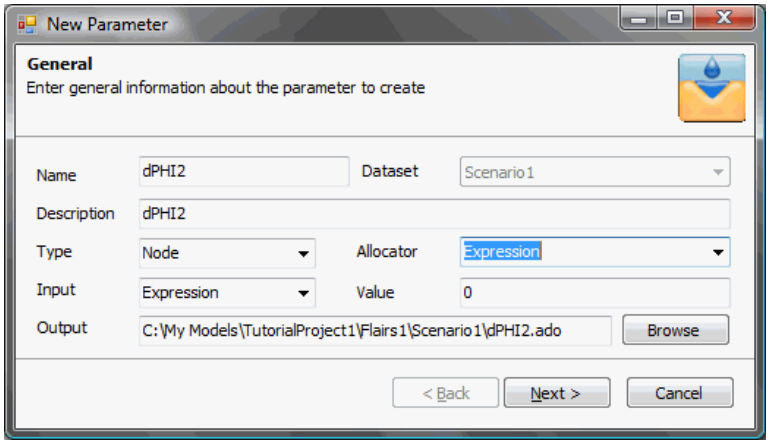
You can combine and process data files in Triwaco; subtract, add, multiply or divide input data with input, output with output and input with output. The result is a new parameter that is stored in one of the data sets (or a specially created data set).

The parameters defined for processing have to be defined in one of the data sets (except grid and initial data set since no triwaco input file in the adore format (.ado) are present). We will create a parameter that represents the difference between the simulated head in aquifer 2 for Scenario1 and the Simulation1, in other words the change in head when the groundwater abstraction of the well with ID 1 is reduced from -2000 m3/d to 0 m3/d. We will do the same as you may remember from paragraph 6.4.8 where we made results available as

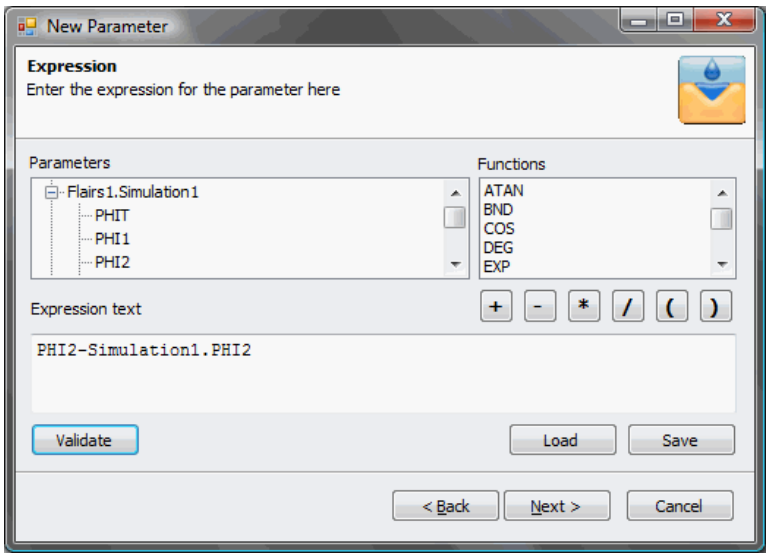
parameter. We showed it for the result parameters **PHIT**, **PHI1** and **PHI2**. It is very easy now to create them for Scenario1. The parameters for Simulation1 are as you may have noticed are shown in the inherited tab. So we simply modify them by which they become parameters part of Scenario1, and will be located in the Parameters tab and also physically present in the dataset directory.

In the same way as before, open the Scenario1 data set. Select the parameter(s) **PHIT**, **PHI1** and **PHI2** that are to be modified with respect to the Simulation1 dataset set by choosing 'Modify' from the context menu. Now the parameters are moved to the Parameters tab. Have for each of these parameters a look at the properties window (select the parameter, 'Parameter', 'Properties'). Select the tab 'Output' and check if the Filename (output file of FLAIRS (Flairs.flo)) is set to the directory Scenario1 (if not the case, browse to the appropriate folder). Since the allocator is None you don't have to do anything further.

Next thing to do is to create a parameter that represents the difference between the simulated head in aquifer 2 for Scenario1 and the Simulation1. Select 'Parameter','New'. In the wizard copy everything from the figure below.

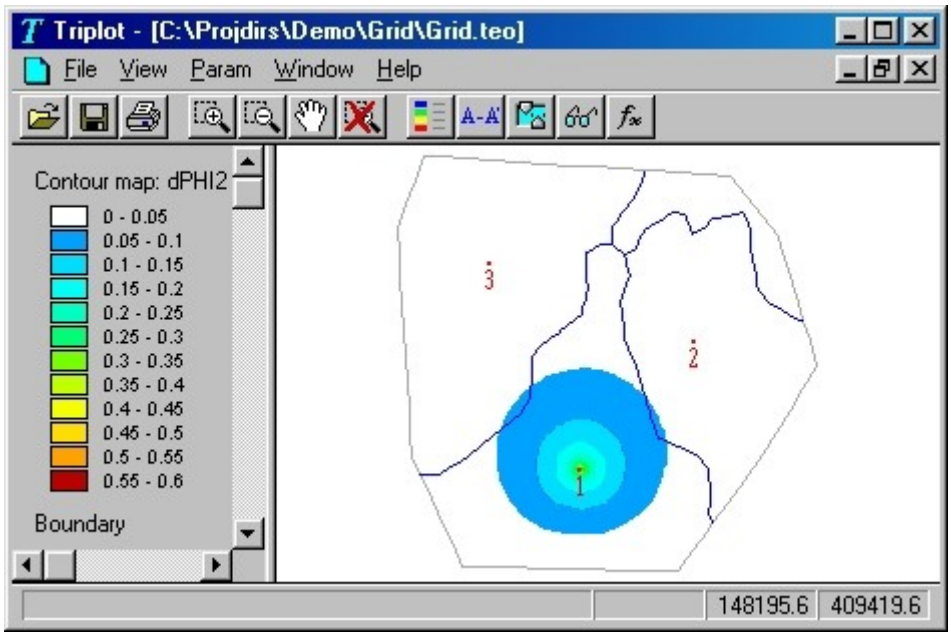


The new parameter is named '**dPHI2**'. Select next and we will now enter the expression as shown below. In the expression the parameter **PHI2** is part of the data set to which the new parameter is added, in this case Scenario1. The parameter **Simulation1.PHI2** means the parameter **PHI2** from dataset Simulation1. So what we have here is that we take the calculated head of aquifer 2 for the Scenario1 simulation and subtract the calculated head of aquifer 2 for the Simulation1 giving the difference **dPHI2** between the two simulations.



Select next and finish. The parameter is now added to the Parameters tab. Allocate the parameter and view the result in TRIPLOT (select 'Param', 'Contour', select 'dPHI2', 'Ok' and set the properties of the contour map). The resulting **dPHI2** may look something like the figure shown below. For other scenario simulations

repeat the above described steps, start by creating a new scenario data set.



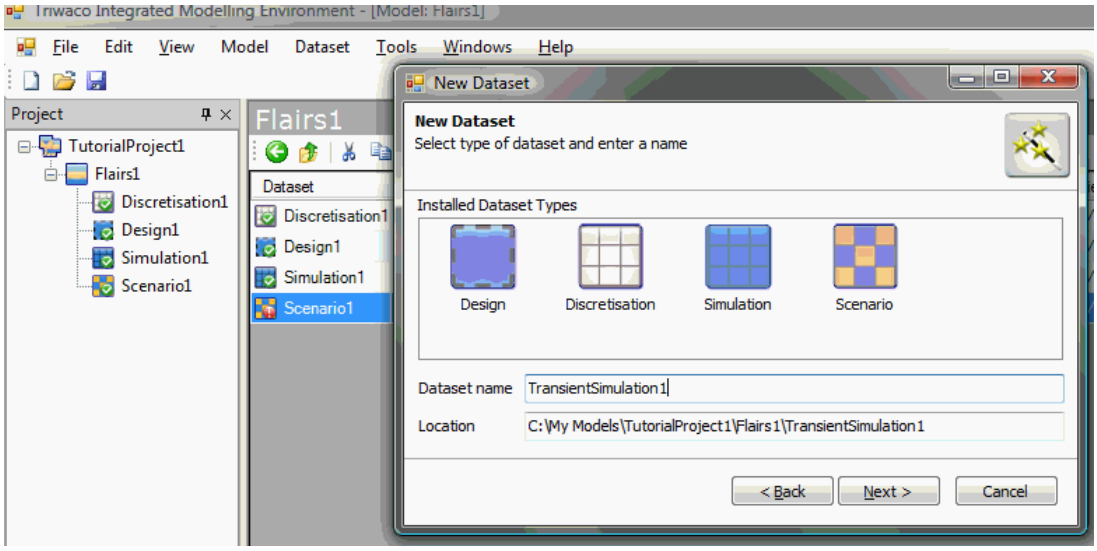
- *The next time you do a simulation run with this scenario the difference is simply obtained by re-allocating the parameter dPHI2.*
- *A full explanation of the application of expressions can be found in the manual. A selection of often used expressions can be found in annex 1a.*

6.6 Setting up a Transient data set

What we have seen until now was creating an calculation grid, creating the conceptual model in the Design dataset and ran the model in the Simulation dataset and also created a Scenario. All of which were simulations in steady-state. Very often however that is not enough and a transient simulation is needed. In this paragraph we will create a transient dataset and make a transient model based on the steady-state model. We will add transient parameters like the storage coefficient and we will also explore two ways of defining parameters that change over the simulation period.

6.6.1 Creating a Transient dataset

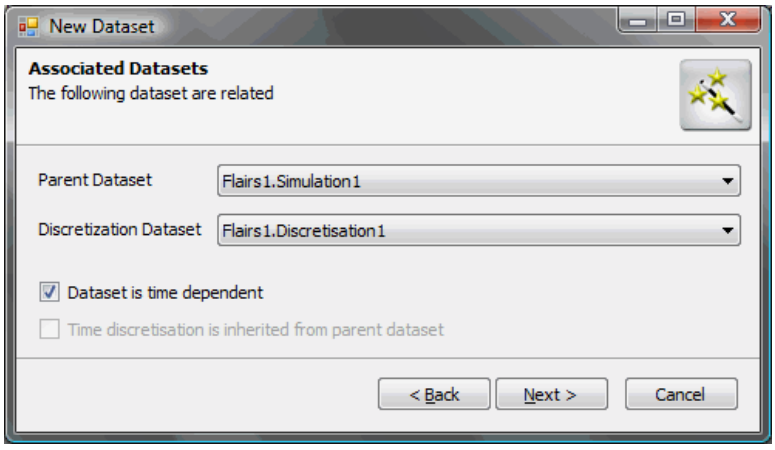
Go back a higher level to the level of the model Flairs. This can be achieved by double clicking on it in the project tree or by opening the context menu of the model Flairs and selecting 'Open'.



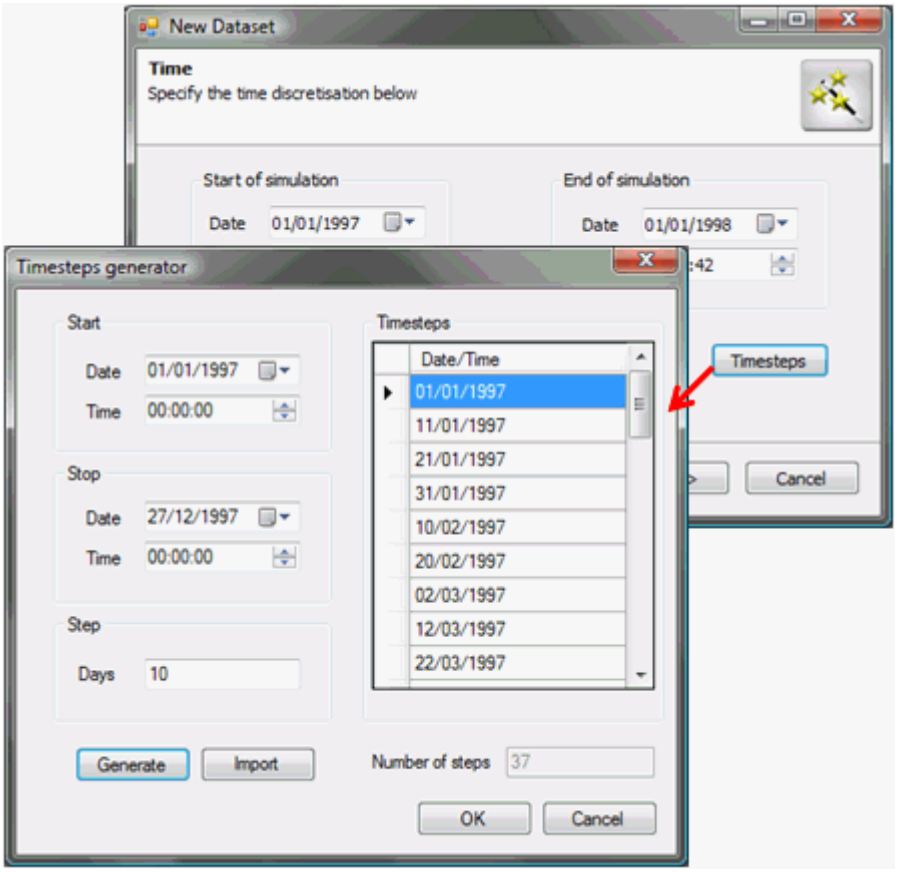
The Transient data set is created by: 'Dataset','Add Dataset'. A pup up window will appear the same as when we created before. Again in the first window select 'Next' to continue. In the second window one can choose the type of dataset. There are four types of datasets each with its own characteristics and purpose:

- **Discretisation**: Defines the calculation grid (boundaries and stresses like watercourses and wells)
- **Design**: Defines the conceptual model using GIS maps and tables.
- **Simulation**: Here is where the data from the conceptual model is linked to the calculation grid. The model is now prepared to run with the model code.
- **Scenario**: Is similar to the simulation dataset. It is base upon the simulation dataset or another scenario dataset. The dataset is created with parameters linked to the parent dataset. Only parameters that need to be altered for that scenario have to be specified.

Since we already have a steady-state model we will create the transient model based upon the existing model. So we will select the 'Simulation' dataset and give it the name TransientSimulation1. Select 'Next'.



In the next window define as the Parent dataset Flairs1.Simulation1 (the steady state model) and of course the existing calculation grid Flairs1.Discretisation1. IMPORTANT is to specify the dataset is Time dependent!!! Select next. Enter the start and end of simulation as given in the figure below. The simulation starts on January 1st 1997 and ends on January 1st 1998. We will use time steps of 10 days. Notice that Triwaco sets the end date to 27th of December because of the time step size of 10 days.



- *It is also possible to import the simulation period and time step size from a txt of CSV file. In that case the time step size can vary throughout the simulated period. For instance when simulation a pumping test. With small time steps in the beginning of the test and larger steps later in the simulation period.*

Select 'Next' and find yourself in the same properties window when defining the design/simulation dataset. Leave everything as it is. All information was inherited from the Simulation1 dataset. Select the 'Advanced' button. The information in the first window is the same as the the Parent dataset Simulation1. Press 'OK' a second window will appear. We will now focus on this window.

There are three tabs. The second and third tab are for more advanced simulations. The tab SF is to activate the SF module. This module can be used to simulate infiltration from lakes or hollows in the dunes and superficial discharge or can be used to simulate surface water flow in the line elements (rivers). The Unsaturated tab can be used to couple the transient groundwater model with an unsaturated zone model, for the definition of groundwater recharge (parameter **RP1**). We will not use these options and they are explained in more detail in other parts of the manual.

The first tab concerns which result parameters will be saved during the simulation. For every aquifer, including the top system, the user has to indicate which parameters he wants to be saved on every time step. This is done under the pull-down menu ‘print option’. You can specify if you want the head in the aquifer, the flux over the top and bottom of the aquifer, and the two-way fluxes to rivers and sources.

At the ‘Points for time lines’ the user can indicate for which locations the change in groundwater head will be followed during the transient calculation. This will result in an ASCII-file, which can easily be imported into Excel for example. This way you can create graphs of groundwater head against time. We will do it in another way.

The maximum head change per time step and the initial time step size are the final two options. The first one is the maximum head change that is allowed per time step. If the head change is larger then the time step size is reduced. The second is the initial time step size. For each time step the model code evaluates how the model converges. If all goes well the time step size is doubled. If not the time step size is halved. So in practical terms it is better to choose a smaller value. In most cases the default values are sufficient.

Close the window and select next until the final window, since we leave everything as it is from the Parent model Simulation1 and finish. The dataset is now created. Notice that the column Regime is transient.

We will now have a look at the input of the different parameters. First the transient parameters constant in time and then to transient parameter that change over time. The latter ones can be defined in two different ways:

- **Specified stress periods**
- **Time series**

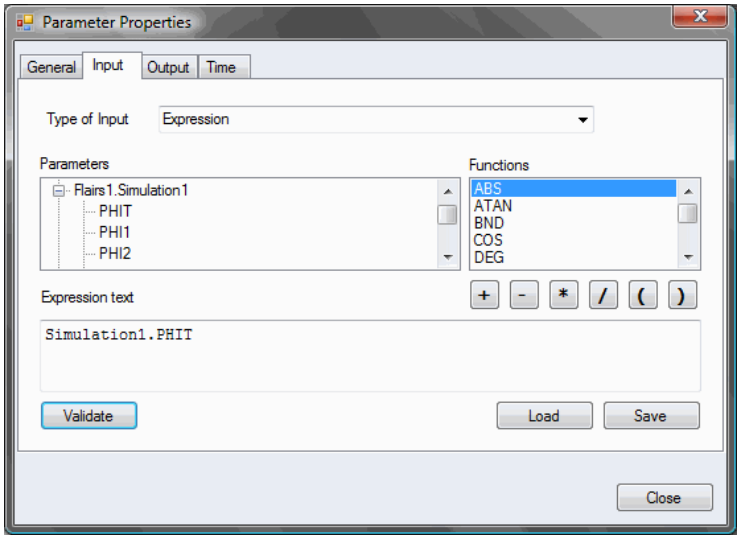
A **specified stress period** is used when a parameter is changed only a few times during the simulated period. For example controlled water level or stopping the abstraction from a well. **Time series allocation** is used for parameters that change with (almost) every time step. For example water levels in a river or precipitation excess. In the following sections we will define all three types.

6.6.2 Input of transient parameters constant in time (initial head, storage coefficient and porosity)

The dataset consists for the most part of inherited parameters of Simulation1 (like a Scenario dataset). However in the Parameter tab there are some new parameters.

The first set are the initial groundwater heads **HT**, **HH1** and **HH2**. Perhaps in paragraph 6.4.9 you created them already to speed up the simulation. In that case they are located in the Inherited tab and you have to move them to the Parameter tab (context menu 'Modify').

For each of them go to the 'Properties window', 'Input' and define by an expression the initial head, which is the calculated head of the steady state Parent model: Simulation1.



You are now back in the dataset. The other two new parameters are the storage coefficient (**SC1** and **SC2**) and effective porosity (**PE**). These parameters are given a constant value. As done and explained before directly enter a value of 0.001 for the **SC1** and **SC2** (elastic storage coefficient) and a value of 0.3 for **PE** (porosity of phreatic aquifer 1). Click refresh (refresh icon) to show if the status bullets are green.

Allocate all six parameters.

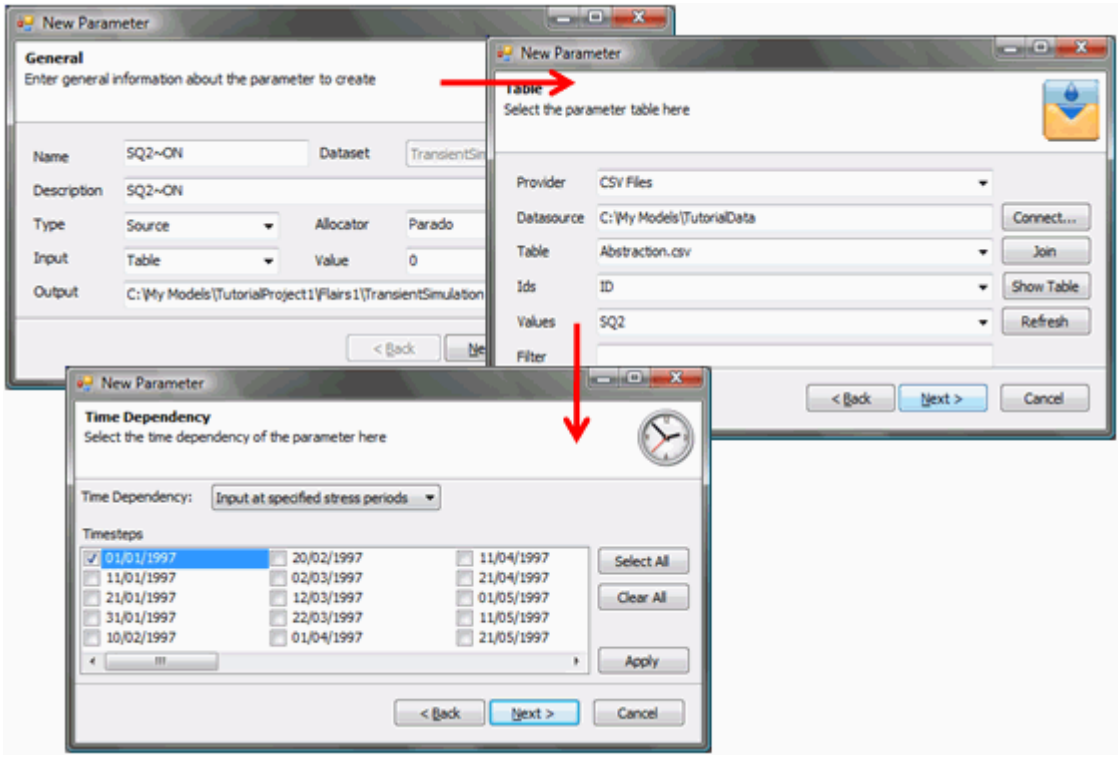
6.6.3 Input of specified stress periods (abstraction well)

To illustrate a specified stress period in the model we will shut down the abstraction well number 1. This is in fact the same calculation as in the formerly described Scenario1. This time however we will look at the change in water level through time.

By defining the input on the specified stress input periods (same periods of 10 days as the time steps defined above), a parameter can be switched on and off. In this way the abstraction rate before and after shutting down the well can be defined. This can be done by the use of a tilde (~) within a parameter name. Triwaco will not use the information, which is behind this tilde, but it will enable you to maintain different values for different periods of time for the same parameter.

Define two different abstraction rates by adding two new parameters: **SQ2~ON** and **SQ2~OFF**. In this case **SQ2~ON** is identical to **SQ2** used in the steady state calculation. **SQ2~OFF** contains an abstraction rate of 0 for well number 1 since it will be shut down. By indicating the valid stress period for both parameters, Triwaco will tell the model code FLAIRS to read **SQ2~ON** for the parameter **SQ2** for the situation whereby the well is still in use and **SQ2~OFF** is read for **SQ2** when the well is shut down (not active).

First we will create the parameter **SQ2~ON**. This is done in three steps of which the first two you have done before. Select 'Parameter','New'. In the wizard copy everything from the 1st window shown below.

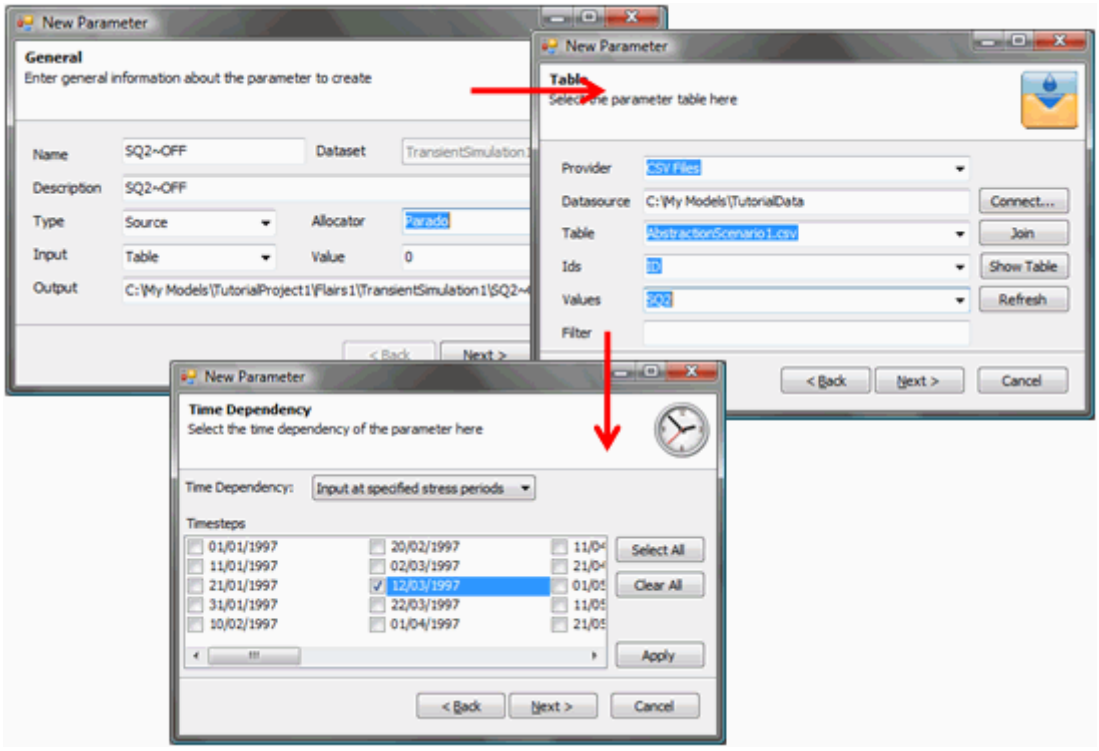


Then select next and follow the instructions for the 2nd window select Type of Input: Table. Then in the Provider field select Microsoft Excel. Then select 'Connect' and in the next window select 'Browse' and locate and select the file AbstractionAq2.xls in the directory My Models\TutorialData\. You can leave the other fields as they are, they are updated automatically by Triwaco. Close this window. You are now back in the 2nd window. Select the Table (in this case worksheet) with abstraction rates that is 'Abstraction aq 2$'. To check if the correct sheet is selected select 'Show Table'. This comes in very handy since it helps to identify the fields ID and Values. 'Close' this window and select the fields as shown in the figure above. Select 'Next'.

You are now in the 3rd window. Select 'Input as specified stress periods', the periods will appear for which we have to specify the specified stress periods. Mark the time steps for which the **SQ2~ON** will be active. You only have to mark the first time step from which on the parameter is active. **SQ2~ON** is active until it is deactivated by the activation of **SQ2~OFF**. So only select 01/01/1997 as shown in the 3rd window above. Select 'Next' and 'Finish'.

- *As you may have noticed the parameter **SQ2~ON** identical to **SQ2**. However **SQ2** will be overruled by the transient parameter.*
- *Since **SQ2~ON** is the same as **SQ2** the parameter can also be created with an expression where the expression is : **Simulation.SQ2**.*

Second we will create the parameter **SQ2~OFF**. This is done in three steps the same way. Select 'Parameter','New'. In the wizard copy everything from the 1st window shown below.



Then select next and follow the instructions for the 2nd window select Type of Input: Table. Then in the Provider field select Microsoft Excel. Then select 'Connect' and in the next window select 'Browse' and locate and select the file AbstractionAQ2.xls in the directory My Models\TutorialData\. You can leave the other fields as they are, they are updated automatically by Triwaco. Close this window. You are now back in the 2nd window. Select the Table with abstraction rates that is 'Abstraction aq 2$'. To check if the correct sheet is selected select 'Show Table'. This comes in very handy since it helps to identify the fields ID and Values. Close this window and select the fields as shown in the figure above (Select 'ID' for field ID and select 'SQ2Scen1' for field Values). Select 'Next'.

You are now in the 3rd window. Select 'Input as specified stress periods', the periods will appear for which we have to specify the specified stress periods. Mark the time steps for which the **SQ2~OFF** will be active. You only have to mark the first time step from which on the parameter is active. **SQ2~OFF** is active until it is deactivated by another parameter. In this simulation we want to shut down the abstraction from well wit ID 1 for the remainder of the simulation period. So only select 12/03/1997 as shown in the 3rd window above. Select next and finish.

Both parameters are now defined. You may now allocate them.

6.6.4 Input is variable through time, definition by time series (precipitation excess)

To illustrate the use of input that is variable through time we will use the most common parameter that is defined in this way, precipitation excess or groundwater recharge.

The precipitation excess is defined by the parameter **RP1** (we used that one when setting up the Design dataset). The **RP1** is present in the inherited tab and is still a parameter that is constant in time. To change it

into a parameter variable in time we have to modify it.
Select the parameter **RP1** choose 'Modify' from the context menu. Now the parameter is moved to the Parameters tab. Go to the Parameter tab and select the parameter **RP1** once more. Open the context menu and open the Properties window.

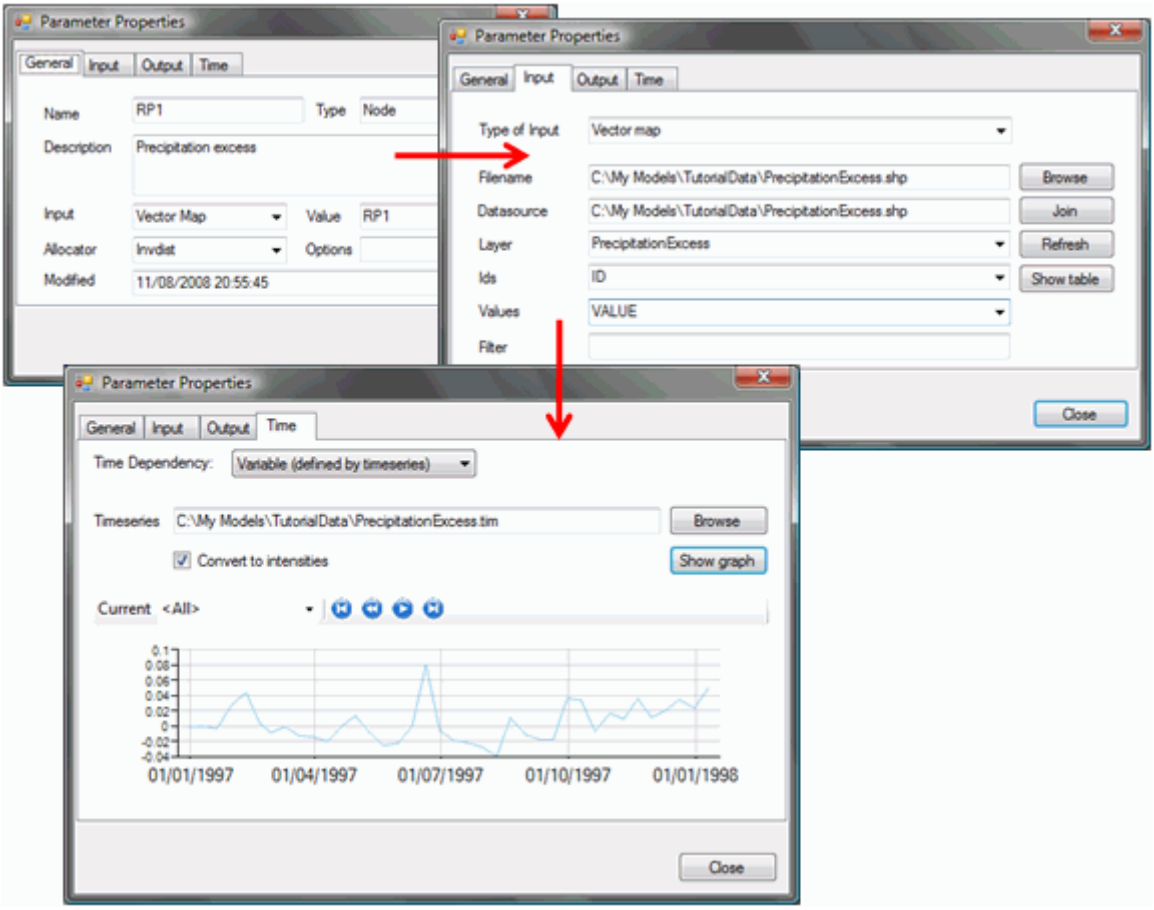
Fortunately we have precipitation excess values from a gauging station in the area. Since it is a one point data set we first, in the tab 'General', change the allocator from Arpadi (allocator for polygons) to InvDist. InvDist uses the inverse distance weighing method. But since it is only one point there will be no interpolation and all values will be the same for all nodes/cells in the model.

In the second tab 'Input' define the vector map shown in the figure below. Select 'Browse' and locate and select the file PrecipitationExcess.shp in the directory My Models\TutorialData\. The fields Filename and Datasource should be changed will now be the same as shown in the figure. Next thing to do is to define the data that is to be extracted from the vector map. Select the same fields as shown below. Leave the file for filter open for now. We have now again defined a vector map. Next thing is to define parameter values variable in time, which is done by using a time series file.

Got to the tab 'Time', where we can define the time series. The time series for a given parameter have to be placed in a **.tim** file with a specified format. An example of a .tim file is given in annex 3 and is included in the directory My Models\TutorialData\PrecipitationExcess.tim. And can easily be prepared in Excel.

The time steps in the .tim file make up stress periods for the values defined in these series. The standard way is that the parameter on each time step within a stress period is assigned the value specified in the .tim file. It is IMPORTANT to note that dates and times defined in the .tim file not necessarily have to correspond to time steps defined in the transient simulation. Triwaco will interpolate.

Now select if not yet selected the Time dependency: Variable. Select 'Browse' and locate and select the file PrecipitationExcess.tim in the directory My Models\TutorialData\. You can choose to either convert the values of the time series to intensities or to values to be converted to averages (see text box). In this case we will use precipitation excess **RP1** by converting time series to intensities, since precipitation is in m/day.



Try the Show Graph to preview the time series. Now close the Parameter Properties. You are back in the dataset and ready to allocate the final transient parameter. Allocate the parameter. This may take some time since it has to allocate for the entire simulation period.

Concert to averages or intensities
Another possibility is to convert to averages. In that case the time dependent values are averaged and appointed to each time step. If you have the following .tim-file

Date Time Value parameter
01/01/2002 00:00 5
11/01/2002 00:00 10

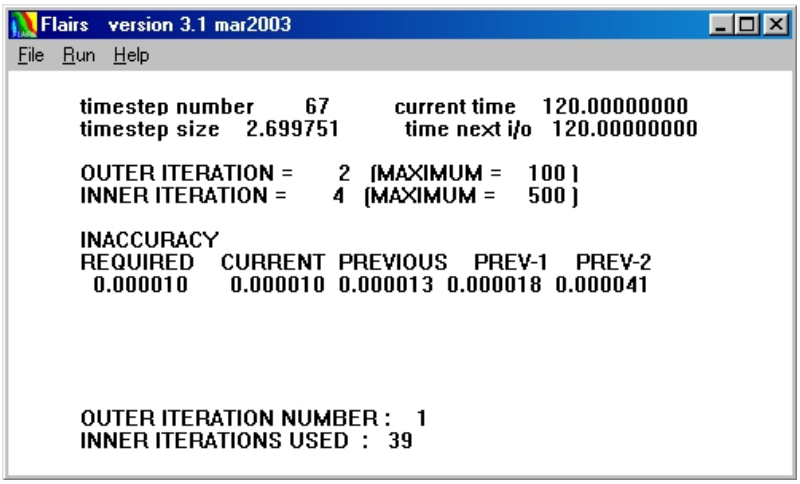
Then for calculation time t=2 days, corresponding with the date 03/01/2002, the value of the parameter is 6.

IMPORTANT to remember is that conversion to intensities is used for parameters like precipitation excess (units X(meters) per time). Conversion to averages is used for parameters like water levels (units X(meters)).

6.6.5 Transient simulation

To run the transient simulation choose 'Dataset','Generate' (creates a model code input file, based on the transient parameters in the dataset and the parameters referenced to the Simulation1 dataset) and then 'Dataset','Run'. A simulation window is opened showing the progress of the simulation. Be aware that a transient simulation takes some time. Flairs will show a window which shows the progress of the simulation.

When ready it closes. You can now view the results with 'Dataset','View','Output'. This will load all results of the simulation into TRIPLOT. The next paragraphs give a quick reference on how to view transient results. For more comprehensive explanation is referred to the part of the manual about TRIPLOT.


Flairs version 3.1 mar2003
File Run Help
timestep number 67 current time 120.00000000
timestep size 2.699751 time next i/o 120.00000000
OUTER ITERATION = 2 (MAXIMUM = 100)
INNER ITERATION = 4 (MAXIMUM = 500)
INACCURACY
REQUIRED CURRENT PREVIOUS PREV-1 PREV-2
0.000010 0.000010 0.000013 0.000018 0.000041
OUTER ITERATION NUMBER : 1
INNER ITERATIONS USED : 39

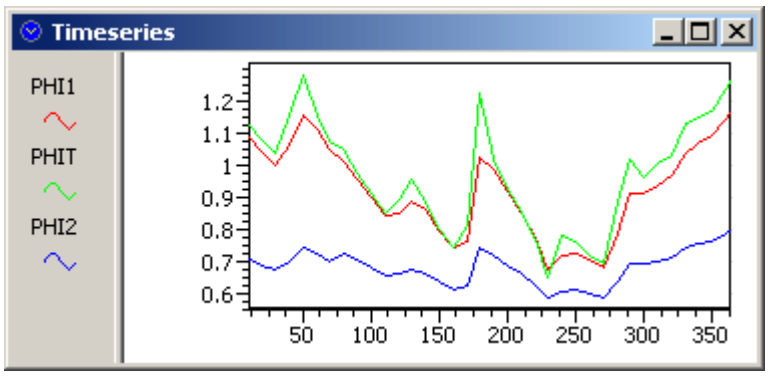
6.6.6 Viewing results

Viewing the results works the same way it did in the other calculation data sets. With the difference that the flairs output file contains information for every stress period. The results may be contoured or classified for any individual stress period. Results may also be presented using time series and animation.

Creating time series

A time series graph can be created provided that transient parameters or transient simulation results are loaded. To create a time series graph 'Time', 'Time series' from the menu bar. Select parameters for time series. The time series for each selected parameter is loaded from the transient file. Then point and select the location (click with the left mouse button in the map) to create the time series graph. To create another graph for another location simply point and select that location. The current time series graph will be refreshed.

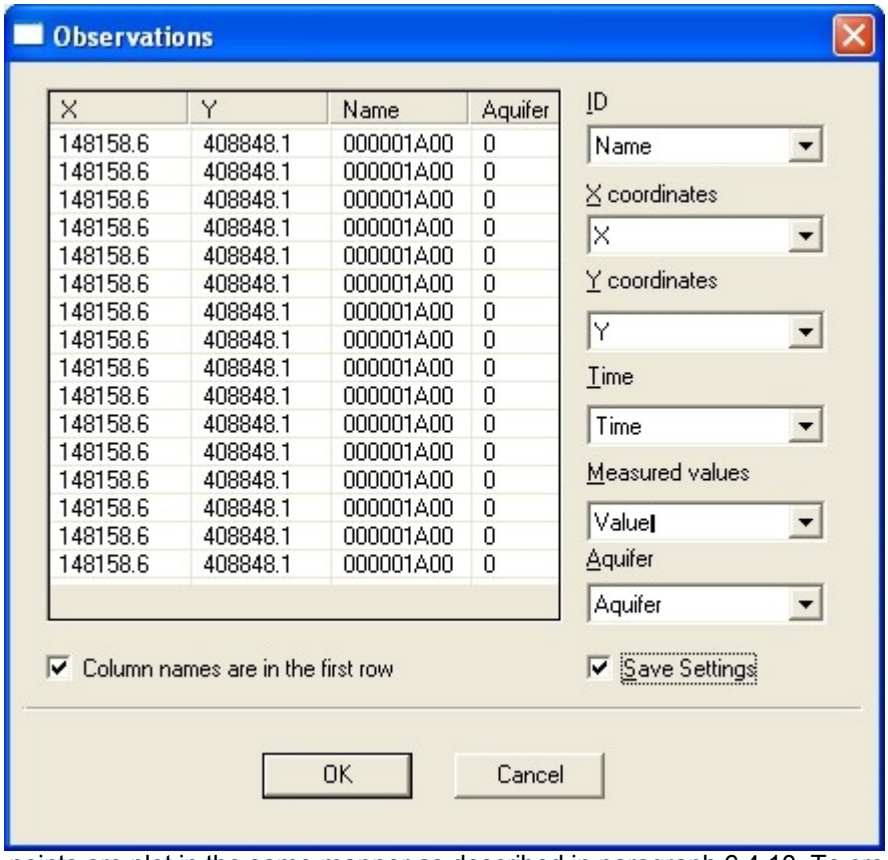
***Note** that, more than one time series graphs can be opened with different parameters.*



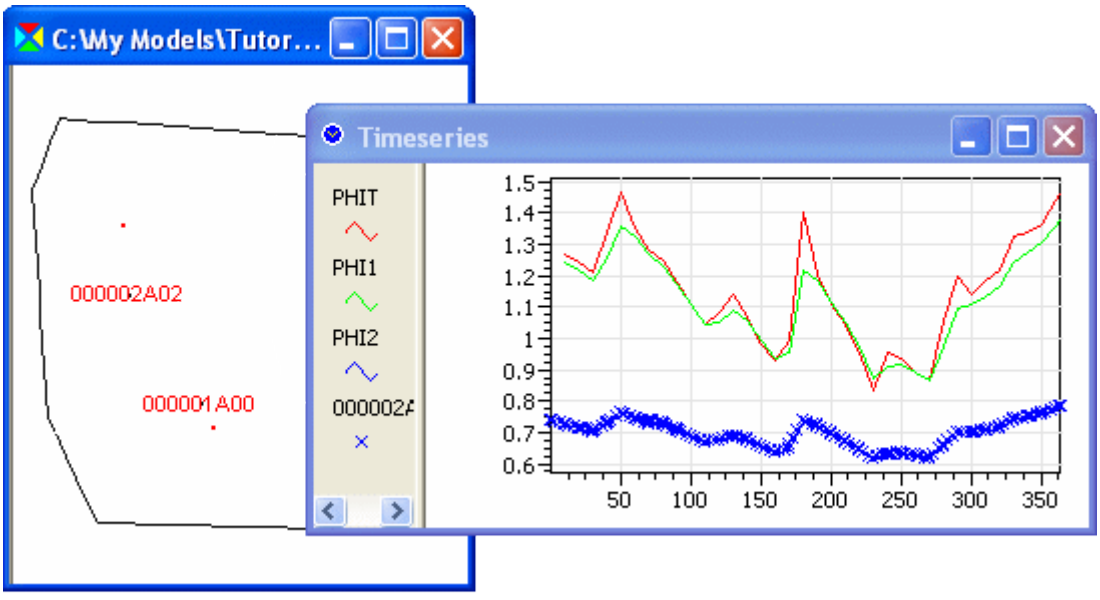
Often however you also want to compare the results for interpretation purposes or processing.

Comparing time series with measured values

In addition to the before described manner to visualize time series we will now add measured values. To load observations 'Time', 'Observations' from the menu bar. A file observations.csv is available in the directory My Models\TutorialData\. Indeed the observations can simply be created in Excel and saved as a comma delimited file.



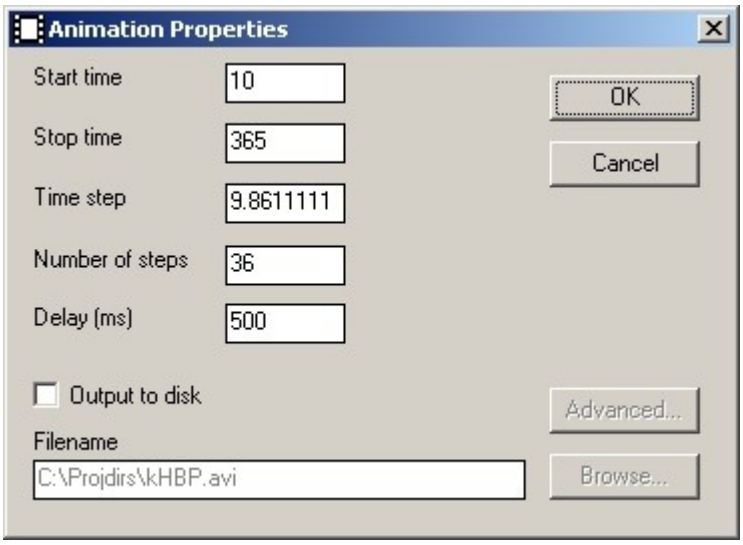
The observations points are plot in the same manner as described in paragraph 6.4.10. To create a time series graph 'Time', 'Time series' from the menu bar. Select parameters for time series. The time series for each selected parameter is loaded from the transient file. Then point and select near the observation to create the time series graph. To create another graph for another location simply point and select that location. The current time series graph will be refreshed.



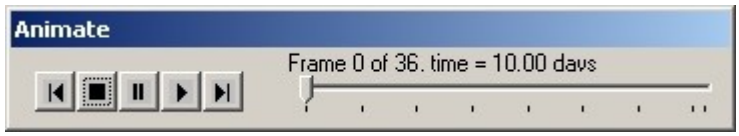
Creating an animation

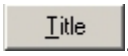
An animation can be created provided that transient parameters or transient simulation results are loaded. To create an animation first create a contour or classified map of the parameter used in the animation. Then

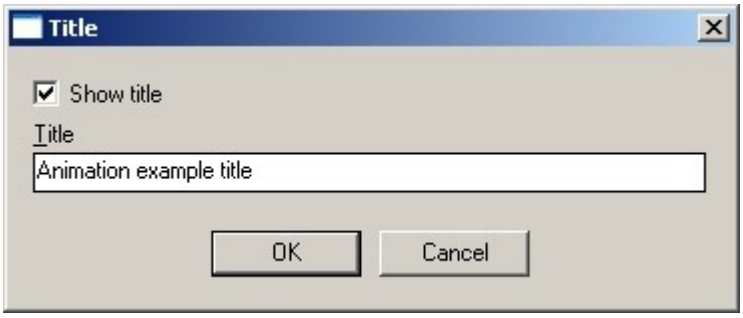
select 'Time', 'Animate' from the menu bar (or simply). The following dialog box will appear.



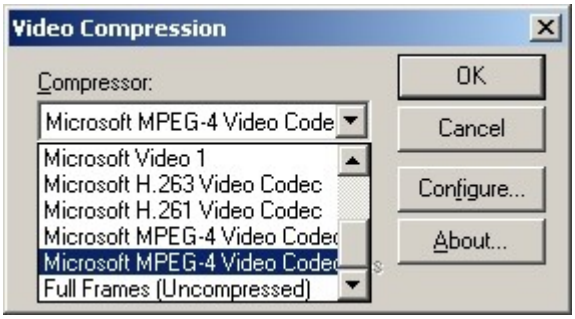
Specify the start time, stop time, time step or number of steps. Delay is used to set the delay time for the frames. Close the dialog box by OK. To start the animation push the play button in the following box. One may also use the time bar to show individual frames.



To create a title and to insert dates in each frame first define the start time by 'Time', 'Starting date' . One is prompted to define date and starting time. Next step is to open the properties window which can be accessed selecting 'Properties' from the 'View' pull-down menu, or right-mouse-button. Select  . The following dialog box will appear. Check 'Show title'. By adding title to the view the date progress is also added to the view. The title will appear at the bottom left and the time progress at the bottom right (often behind the frame dialog box).



An animation may also be saved to disk by selecting output to disk. Various file compression formats are available.


Video Compression
Compressor:
Microsoft MPEG-4 Video Code
Microsoft Video 1
Microsoft H.263 Video Codec
Microsoft H.261 Video Codec
Microsoft MPEG-4 Video Codec
Microsoft MPEG-4 Video Codec
Full Frames (Uncompressed)
OK
Cancel
Configure...
About...

Annex 1a Application of the parameter allocators

Type of data	Use Allocator
Parmeters covering the whole model ('node values')	
- Input as a constant value	Constant
- Input by polygons	Arpadi, Warp, Kriging
- Input by point values	InvDist, Arpadi, Kriging, etc.
- Input by (a large amount of) point values	Tin, Kriging
Parameters for linear surface water (type 'river')	
- Input as a constant value	Constant
- Input by linked points	ParRiv
- Input by polygons	Arpadi, Warp, Kriging
Parameters for sources (type 'source')	
- Input as a constant value	Constant
- Input by point values	SrcParado
- Input by polygons	Arpadi
Parameters for boundary conditions (type 'boundary')	
- Input as a constant value	Const
- Input by linked points	ParBou
- Input by polygons	Arpadi
Input based on one or more other parameters	Expression

Application of expressions

The Expression allocator evaluates an expression and calculates (creates) a new ***Adore***-block. An expression may contain set-names, numbers, functions, factors and operators. Three types of operators may be distinguished: mathematical operators, relational operators and logical operators.

Definition	Description
Set-names	Parameter names as defined in **Triwaco**, consisting of a combination of alphanumeric characters. The parameter may be preceded by the name of one of the project's data sets and a $-sign: e.g., cal$TX1
Numbers	integer and real numbers: e.g., 15, -0.456
Factors	Consist of numbers, expressions, functions or identifiers.
Mathematical operators	+, -, * and /
Relational operators	>, ≥ **(>=)**, = **(==)**, ≤ **(<=)** and <
Logical operators	'AND' ('**&&**'), 'OR' ('**||**') and 'NOT' ('**=!**') and 'IF' 'THEN' ('**?**') and 'ELSE' ('**:**')

Functions	(simple) mathematical functions:	
	abs(x)	Returns the absolute value of '***x***'
	atan(y,x)	Returns the arc tangent of ('***y/x***')
	BND(x)	Returns the value of '**x**' at boundary nodes
	cos(x)	Returns the cosine of '***x***'
	deg(x)	Converts radians ('***x***') to degrees
	exp(x)	Returns the value of ***e*** raised to the power '***x***'
	IF(x,y,z)	Evaluates the logical expression: IF ('***x***') THEN ('***y***') ELSE ('***z***') Equivalent to the expression: ('***x***')?('***y***'):('***z***')
	ln(x)	Returns the natural logarithm of '***x***'
	log(x)	Returns the 10 log of '***x***'
	max(x,y)	Returns the largest value of '***x***' and '***y***'
	min(x,y)	Returns the smallest value of '***x***' and '***y***'
	NODE(x)	Returns the value of '***x***' at **all Nodes**; if the value of '***x***' does not exist at a Node a zero value (**0**) is assumed
	rad(x)	Converts degrees ('***x***') to radians
	RIV(x)	Returns the value of '***x***' at river nodes
	sign(x)	Returns the sign of '***x***' (-1, 0 or +1)
	sin(x)	Returns the sine of '***x***'
	sqr(x)	Returns the square of '***x***'
	sqrt(x)	Returns the square root of '***x***'
	SRC(x)	Returns the value of '***x***' at source nodes
	tan(x)	Returns the tangent of '***x***'

Important note: The setname or data set name should NOT contain an underscore (data_set$set_name).

Examples of expressions
In the following table examples of the more or less frequently used expressions are listed.

PHIT	adore block with values equal to those of the set with the matching set name: 'PHIT'
Result$PHI1	adore block with values equal to those of set 'PHI1' belonging to the data set with the name: 'result'
12	adore block with the constant value 12
PHI1-PHIT	adore block with values equal to (PHI1 - PHIT), being the difference of the adore blocks with set names 'PHI1' and 'PHIT' respectively
QRCH>0	Boolean adore block containing **integer** values: equal to **1** where QRCH > 0 and equal to **0** where QRCH <= 0
(PHI1-PHIT) * (QRCH>0 && QKW1>0)	Real adore block containing values equal to **0** where QRCH <= 0 or QKW1 <= 0 and to (**PHI1-PHIT**) where both QRCH > 0 and QKW1 > 0
(RL1>TH1)?RL1:(TH1 + 0.01)	Real adore block containing values equal to **RL1** where RL1 > TH1 and to (**TH1+0.01**) where RL1 <= TH1
IF(RL1>TH1,RL1,TH1+0.01)	Real adore block containing values equal to **RL1** where RL1 > TH1 and to (**TH1+0.01**) where RL1 <= TH1
sqrt(log(cos(TX1*TH1)+1)	adore block that contains values equal to the results after evaluating the expression: $\sqrt{\log\left(\cos\left(TX1 * TH1\right) + 1\right)}$
QRI1/AREA	Specific river flux in m/d (river flux divided by node influence area)
MIN(PHIT,RP13)	Minimum value of **PHIT** and **RP13**: cut off **PHIT** at surface level
PHIT > RP13 ? RP13 : PHIT	Same as above
IF(PHIT>RP13, RP13, PHIT)	Same as above

Note:
Using Boolean expressions the result set will contain **integer** values if the expression starts with the Boolean expression and will contain **real** values if the Boolean expression is preceded with a (real) value or another expression.

Thus: (PHI1-PHIT) * (QRCH>0 && QKW1>0) results in a **real** Adore set and
(QRCH>0 && QKW1>0) * (PHI1-PHIT) results in an **integer** Adore set.

Annex 3b Proposed default parameter values for demo-model

IR	Type of top system	11
RP1	Precipitation excess	0.001 m/day (See description in text)
RP2	Resistance semi-pervious layer	20 days
RP3	Drainage resistance	250 days
RP4	Infiltration resistance	900 days
RP5	Drainage level	See description in text
RL1	Top aquifer 1	(see MV25.ung en MV25.par)
TH1	Base aquifer 1	-10.0 m
PX1	Permeability aquifer 1	25 m/day
TX2	Transmissivity aquifer 2	3500 m^2/day
CL1	Resistance aquitard 1	250 day
IB1, IB2, etc.	Type of Boundary condition	0 (fixed head)
BH1, BH2, etc.	Boundary Head	0.50 m
BA1, BA2, etc.	Boundary condition for flux	0 (only when IB1=1)
BB1, BB2, etc.	Boundary condition for flux	0 (only when IB1=1)
IS1, IS2, etc.	Type of Abstraction	0 (fixed discharge)
SQ1, SQ2, etc.	Discharge amount	(see text, - values means abstraction)
SH1, SH2, etc.	Fixed head in discharge well	0 (only when IS1=1)
RA1	River activity	1 (all rivers active)
HR1	Water level	0 (define with linked points, see text)
RW1	River width	15 m
CD1	Drainage resistance river	5 days
CI1	Infiltration resistance river	25 days

Annex 2 Lay out calibration file (measured heads)

The file with the measured heads should have the following lay out:

```
[title]
[code well 1] [x-co-ordinate] [y-co-ordinate] [cluster no.] [no. aquifer] [measured head]
[code well 2] [x-co-ordinate] [y-co-ordinate] [cluster no.] [no. aquifer] [measured head]
[code well 3] [x-co-ordinate] [y-co-ordinate] [cluster no.] [no. aquifer] [measured head]
[code well 4] [x-co-ordinate] [y-co-ordinate] [cluster no.] [no. aquifer] [measured head]
... etc.
```

For phreatic heads you fill in a 0 for the number of the aquifer. With the cluster number one can assign certain wells to a specific group for statistical analyses.

The well data has to be entered in a **fixed format**:

[A10] [F10.*] [F10.*] [I5] [I5] [F10.*]

where	[A10]	a text with a maximum of 10 characters (incl. Spaces)
	[F10.*]	a number with a decimal point, 10 characters long
	[I5]	an integer number (without a point), 5 characters long

The name of the file with calibration values is by definition **calib.chi**

An Example (spaces are designated as '~'):

```
measurements 1995
~~52AP0014~142057.00~325995.00~~~~1~~~~1~~~~~24.01
~52AP0014~~~142057.0~~325995.0~~~~1~~~~2~~24.52~~~
52AL0133~~~~~145239.~~~323963.~~~~1~~~~0~~~~25.11~
52~AP~1055~~137224.0~~321401.0~~~~1~~~~1~~~~23.980
```

Annex 3 Lay out time series files

Below an example of a RP1.tim file is given. In the first column the date is stated. In the second one the time. In the final column the parameter value is given for the period before the stated moment of time stated on that particular row.

```
1
31/12/1996 00:00 -0.001
10/01/1997 00:00 -0.00055
20/01/1997 00:00 -0.00285
31/01/1997 00:00 0.02875
10/02/1997 00:00 0.0439
20/02/1997 00:00 0.00365
28/02/1997 00:00 -0.0084
10/03/1997 00:00 -0.00095
20/03/1997 00:00 -0.0121
.
.
20/12/1997 00:00 0.03475
31/12/1997 00:00 0.0234
10/01/1998 00:00 0.05
END
END
```